

**US Army Corps
of Engineers**
Waterways Experiment
Station

COSFLOW: A Finite Element Model Coupling One-Dimensional Canal, Two-Dimensional Overland, and Three-Dimensional Subsurface Flow

by Gour-Tsyh (George) Yeh, Jing-Ru (Ruth) Cheng, Ming-Hsu Li,
Hwai-Ping (Pearce) Cheng, Pennsylvania State University

Hsin-Chi J. Lin, WES

[illegible]

Approved For Public Release; Distribution Is Unlimited

19971008 097

DTIC QUALITY INSPECTED 3

The contents of this report are not to be used for advertising, publication, or promotional purposes. Citation of trade names does not constitute an official endorsement or approval of the use of such commercial products.

The findings of this report are not to be construed as an official Department of the Army position, unless so designated by other authorized documents.



PRINTED ON RECYCLED PAPER

COSFLOW: A Finite Element Model Coupling One-Dimensional Canal, Two-Dimensional Overland, and Three-Dimensional Subsurface Flow

by Gour-Tsyh (George) Yeh, Jing-Ru (Ruth) Cheng, Ming-Hsu Li,
Hwai-Ping (Pearce) Cheng

Department of Civil and Environmental Engineering
Pennsylvania State University
University Park, PA 16802

Hsin-Chi J. Lin

U.S. Army Corps of Engineers
Waterways Experiment Station
3909 Halls Ferry Road
Vicksburg, MS 39180-6199

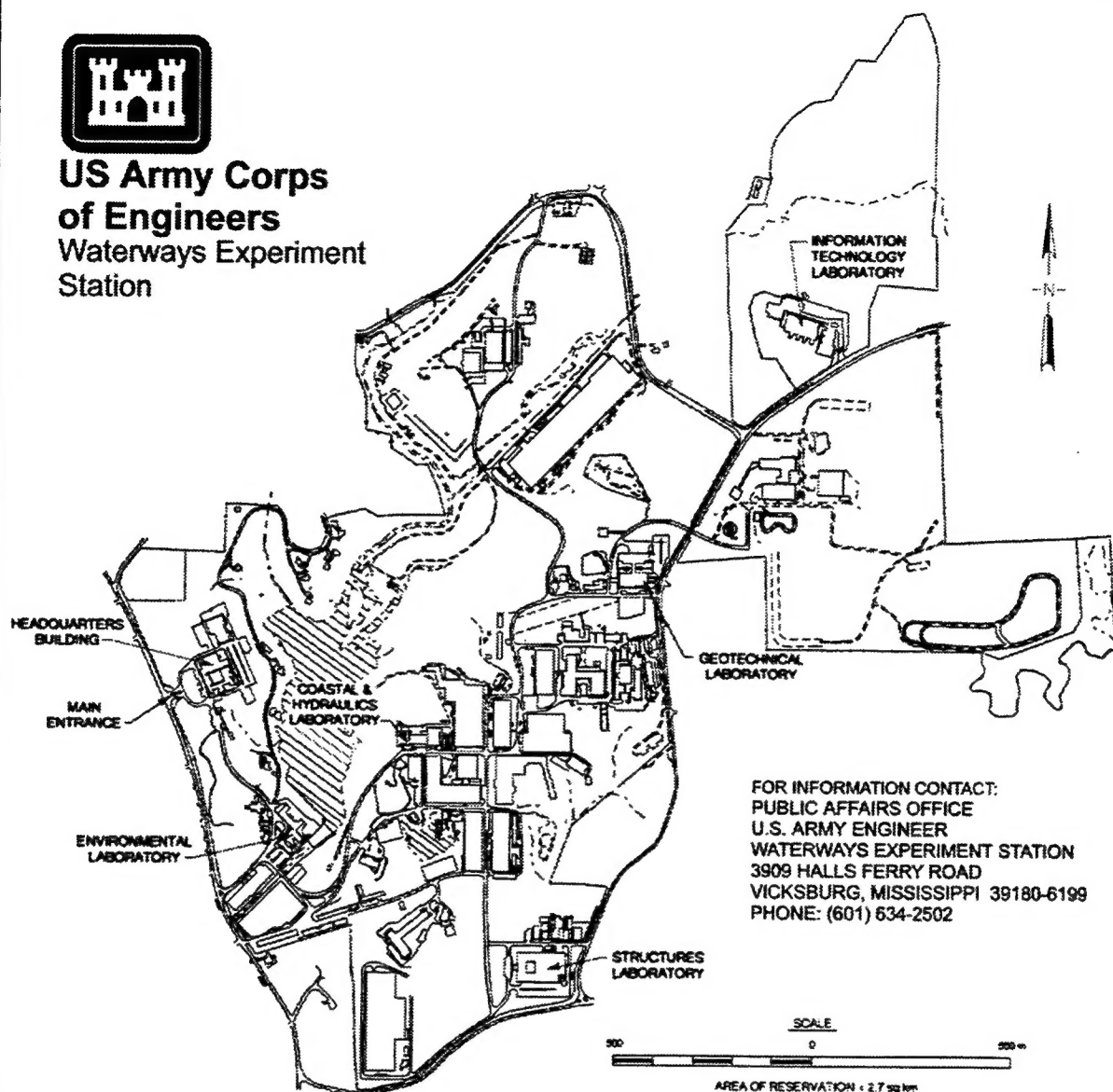
Final report

Approved for public release; distribution is unlimited

DTIC QUALITY INSPECTED 3



**US Army Corps
of Engineers**
Waterways Experiment
Station



FOR INFORMATION CONTACT:
PUBLIC AFFAIRS OFFICE
U.S. ARMY ENGINEER
WATERWAYS EXPERIMENT STATION
3909 HALLS FERRY ROAD
VICKSBURG, MISSISSIPPI 39180-6199
PHONE: (601) 634-2502

Waterways Experiment Station Cataloging-in-Publication Data

COSFLOW : a finite element model coupling one-dimensional canal, two-dimensional overland, and three-dimensional subsurface flow / by Gour-Tsyh (George) Yeh ... [et al.] ; prepared for U.S. Army Engineer District, Jacksonville.

198 p. : ill. ; 28 cm. — (Technical report ; CHL-97-20)

Includes bibliographic references.

1. COSFLOW (Computer program) 2. Groundwater flow — Mathematical models. 3. Finite element method. I. Yeh, Gour-Tsyh. II. United States. Army. Corps of Engineers. Jacksonville District. III. U.S. Army Engineer Waterways Experiment Station. IV. Coastal and Hydraulics Laboratory (U.S. Army Engineer Waterways Experiment Station) V. Title: Finite element model coupling one-dimensional canal, two-dimensional overland, and three-dimensional subsurface flow. VI. Series: Technical report (U.S. Army Engineer Waterways Experiment Station) ; CHL-97-20.

TA7 W34 no.CHL-97-20

Contents

Preface	viii
1. INTRODUCTION	1
2. THE COSFLOW PROGRAM STRUCTURE	5
2.1 Purpose of COSFLOW	5
2.2 Description of COSFLOW subroutines	12
3. ADAPTATION OF COSFLOW TO SITE SPECIFIC APPLICATIONS	49
3.1 Parameters Specifications	49
3.2 Soil Property Function Specifications	63
3.3 Specification of Discharge Rates for Gates and Weirs	65
3.4 Input and Output devices	68
3.4.1 File Organization	68
3.4.2 Super File	70
4. EXAMPLE PROBLEMS	77
4.1 Example 1A: 2-D Overland Flow	77
4.2 Example 1B: 2-D Overland Flow	85
4.3 Example 2: 3-D Subsurface Flow	91
4.4 Example 3: 1-D Channel Water Budget and 3-D Subsurface Flow	99
4.5 Example 4: 2-D Overland Flow and 3-D Subsurface Flow	113
4.6 Example 5: 1-D Channel Water Budget, 2-D Overland Flow, and 3-D Subsurface Flow	122
APPENDIX A: DATA INPUT GUIDE FOR COSFLOW	135
A.1 Meshes	135
A.2 Model file for 3-D subsurface flow and transport	141
A.3 Model file for 1-D canal flow	151
A.4 Model file for 2-D overland flow and retention pond	156
A.5 The formats of initial condition files and solution files	165

APPENDIX B: NUMERICAL FORMULATION	172
B.1 Numerical Approximation of the 1-D Channel Water Budget Equation	174
B.1.1 Computation of Interaction Between Subsurface and Channel Flow Systems	175
B.2 Numerical Approximation of the 2-D Overland Systems . . .	177
B.2.1 Numerical Approximation of 2-D Overland Flow Equation	178
B.2.1.1 Spatial discretization with the Galerkin finite element method	178
B.2.1.2 Base and weighting functions	181
B.2.1.3 Numerical integration	182
B.2.2 Numerical Approximation of Retention Pond Water Budget Equation	184
B.2.3 Computation of Interaction between Subsurface Flow and Overland Flow System	186
B.2.4 Computation of Interaction between Subsurface Flow and Retention Pond Water Budget	187
BIBLIOGRAPHY	188
SF 298	

List of Figures

Figure 2.1	Program structure of COSFLOW	14
Figure 2.2	Program structure of subroutines FILES, RINPUT1, RPINP2, RINPUT2	15
Figure 2.3	Program structure of subroutines DATA2D and DATA3D	16
Figure 2.4	Program structure of subroutines BUDGET and BGFA	17
Figure 2.5	Program structure of subroutines OF2D and SF3D	18
Figure 3.1	The sketch of a simulation problem for demonstration . .	55
Figure 3.2	1-D canal discretization for demonstration	59
Figure 3.3	Super File Format	71
Figure 4.1	Problem definition of Example 1A	78
Figure 4.2	The overland domain and its discretization for Example 1A	79
Figure 4.3a	Simulated water stage at $t = 100$ sec for Example 1A . .	83

Figure 4.3	Simulated water stage at (b) $t = 5,000$ sec and (c) $t = 60,000$ sec for Example 1A	84
Figure 4.4	Problem definition for Example 1B	86
Figure 4.5	The rating curve for the radiation boundary in Example 1B	87
Figure 4.6a	Simulated water stage at $t = 1,850$ sec for Example 1B	89
Figure 4.6	Simulated water stage at (b) $t = 5,000$ sec and (c) $t = 10,000$ sec of Example 1B	90
Figure 4.7	Problem definition and sketch for Example 2	92
Figure 4.8	Finite element discretization for Example 2	93
Figure 4.9a	Water table for Example 2	96
Figure 4.9b	Water table on the x-z cross section through the pumping well for Example 2	97
Figure 4.10a	Velocity distribution throughout the domain for Example 2	98
Figure 4.10b	Velocity distribution on the x-z cross section through the well for Example 2	98
Figure 4.11	The subsurface domain and discretization for Example 3	101
Figure 4.12	The channel reach nodes for Example 3	102
Figure 4.13	Channel element nodes for Example 3	104
Figure 4.14	Simulated water stage of Reach 1, 6, and 8 for Example 3	110
Figure 4.15	Simulated total head on the top surface of 3-D domain for Example 3	110
Figure 4.16	The subsurface domain and its discretization for Example 4	114
Figure 4.17	The 2-D overland domain and its discretization for Example 4	115
Figure 4.18	Simulated overland water depth of Example 4 at $t = 9000$ sec	118
Figure 4.19	Simulated overland water depth at $t = 16,200$ sec for Example 4	119
Figure 4.20	Simulated subsurface pressure head on the y-z plane across $x = 250$ m at (a) $t = 3,600$ sec, (b) $t = 9,000$ sec, and (c) $t = 18,000$ sec	120

Figure 4.21	The subsurface domain and its discretization for Example 5	123
Figure 4.22	The overland domain and its discretization for Example 5	125
Figure 4.23	The 1-D canal and its relation with the top of the subsurface domain	126
Figure 4.24	Simulated water stage of canal and retention pond	130
Figure 4.25	Simulated water stage of 2-D overland flow for Example 5	130
Figure 4.26	Simulated velocity field of 2-D overland flow at $t = 18000$ sec for Example 5	131
Figure 4.27	Total head contour at $Z = 196$ m of 3-D subsurface at $t = 18,000$ sec for Example 5	131
Figure A.1	Numbering sequence and face ID in a 3-D element	136
Figure A.2	The file format of 3-D geometry file	137
Figure A.3	Numbering sequence in a 2-D element	139
Figure A.4	Geometry file format for 2-D meshes	139
Figure A.5	Text Data Set File Format	166
Figure B.1	A quadrilateral element in local coordinate	182

List of Tables

Table 3.1	Input Files	68
Table 3.2	Output Files	69
Table 3.3	List of filenames, the associated conditional variables, logical unit, file type, variables assigned in COSFLOW, and brief descriptions	72
Table 4.1	Definition of channel reaches for Example 3	103
Table 4.2	Definition of channel elements for Example 3	105
Table 4.3	Channel element nodes and their associated subsurface nodes for Example 3	106
Table A.1	The data input format of geometry file for 3-D elements	137
Table A.2	The data input format of geometry file for 2-D elements	140

Table A.3	The data input format of model file for 3-D subsurface flow and transport	141
Table A.4	The data input format of model file for 1-D canal flow simulation	151
Table A.5	The data input format of model file for 2-D overland flow simulation	157
Table A.6	The data input format of model file for 2-D retention pond simulation	163

Preface

This report on a finite element computer model coupling one-dimensional canal, two-dimensional overland, and three-dimensional subsurface flow was prepared for the U.S. Army Engineer District, Jacksonville.

The study was conducted as part of the South Florida Groundwater Modeling study in the Hydraulics Laboratory (HL) of the U.S. Army Engineer Waterways Experiment Station (WES) from 1995 to 1997 under the general direction of Messrs. F. A. Herrmann, Jr., Director, HL; R. A. Sager, Assistant Director, HL; and Dr. William D. Martin, Acting Chief, Hydro-Science Division (HD), HL.

The report was prepared by Drs. Gour-Tsyh Yeh, Jing-Ru Cheng, Ming-Hsu Li, and Hwai-Ping Cheng, Pennsylvania State University, University Park, PA; and Dr. Hsin-Chi J. Lin, Watershed Systems Group, Hydro-Sciences Division.

This report is being published by the WES Coastal and Hydraulics Laboratory (CHL). The CHL was formed in October 1996 with the merger of the WES Coastal Engineering Research Center and Hydraulics Laboratory. Dr. James R. Houston is the Director of the CHL and Messrs. Richard A. Sager and Charles C. Calhoun, Jr., are Assistant Directors.

At the time of publication of this report, Director of WES was Dr. Robert W. Whalin.

The contents of this report are not to be used for advertising, publication, or promotional purposes. Citation of trade names does not constitute an official endorsement or approval for the use of such commercial products.

1. INTRODUCTION

COSFLOW is used to mainly investigate the interaction between 3-D subsurface flow, 2-D overland flow incorporating retention ponds, and 1-D channel flow. For the 3-D subsurface flow module, the Galerkin finite element method is used to discretize the Richard's equation; for the 1-D channel flow module and retention ponds, water budget is employed to derive the governing equation; and for 2-D overland flow module, the Galerkin finite element is used to solve the diffusion type continuity equation.

The purpose of this manual is to provide guidance for users of the computer code to enable them to employ the model for site-specific applications. Thus, Section 2.1 lists the governing equations and describes initial and boundary conditions for which COSFLOW is designed to solve. For brevity, the solute transport in the subsurface is not included here but can be found in the manual of 3DFEMFAT [Yeh et al., 1994]. Section 2.2 contains the description of all subroutines in COSFLOW. Again, subroutines for the solute transport are excluded. This section should facilitate the understanding of the code structure by the users. Since occasions may arise that the users have to modify the code, this section should help them to trace the code so they can make necessary adjustments for their purposes. General information on input parameters required by each subroutine is also provided. Section 3.1 contains the parameter specification. For each application, users must assign 151 maximum control-integers (including those for the subsurface solute transport). Section 3.2 describes soil property function specifications such that the users will be able to modify subroutine SPROP3 for each site-specific applications. Section 3.3 describes discharge rate specification of gates and weirs such that the users can modify subroutines QGATE and/or QWEIR for their own purposes. Section 3.4 describes files required for the execution of COSFLOW. Appendix A contains the data input guide that is essential for any site specific application. Appendix B gives the numerical detail in approximating the governing equations.

The users may choose whatever consistent units they want in the input. Units of mass (M), length (L),

and time (T) are indicated in the input description.

The special features of COSFLOW are its flexibility and versatility in modeling a range of real-world problems. The model is designed to, in the 3-D subsurface module, (1) treat heterogeneous and anisotropic media consisting of as many geologic formations as desired, (2) consider both distributed and point sources/sinks that are spatially and temporally dependent, (3) deal with transient Dirichlet boundary conditions, (4) handle time-dependent fluxes due to the gradient of pressure head or concentration varying along the Neumann boundary, (5) treat time-dependent total fluxes distributed over the Cauchy boundary, (6) handle time-dependent river heads for the River boundary (7) automatically determine Variable boundary conditions of evaporation, infiltration, or seepage on the soil-air interface for the flow module and Variable boundary conditions of inflow and outflow for the transport module, (8) include the off-diagonal hydraulic conductivity components in the Richard's equation for dealing with cases when the coordinate system does not coincide with the principal directions of the hydraulic conductivity tensor, (9) give three options for estimating the nonlinear matrix, (10) include three options (successive point iterations, polynomial preconditioned conjugate gradient methods, and incomplete Cholesky preconditioned conjugate gradient methods) for solving the linearized matrix equations, (11) provide two options of treating the mass matrix - consistent and lumping, (12) automatically reset time step size (of 3-D subsurface) when boundary conditions or source/sinks change abruptly, and (13) check the mass balance computation over the entire region for every time step. For the 1-D channel module, the model can (1) automatically adjust time step size (of 1-D channel), based on the allowed water-depth change during one channel time step (for the reason of accuracy) and (2) treat the water flux through the channel-subsurface interface as line, vertical plane, or composite plane sources/sinks to the subsurface system, so that a variety of real world problems can be handled. For the 2-D overland module, the model can (1) allow retention ponds to store the water pumped from canal, (2) compute the seepage or infiltration rate according to the water depth in the pond and the distribution of pressure head in the subsurface,

(3) treat the volumetric flow rate from retention ponds as point sources contributing to overland flow, (4) prescribe head boundary or flux boundary, and (5) provide time-dependent rainfall rate to simulate the real world problem.

One thing should be noted that only transient simulations are considered in this version since the interaction between channel, overland, and subsurface system is always time dependent.

2. THE COSFLOW PROGRAM STRUCTURE

2.1 Purpose of COSFLOW

COSFLOW is designed to solve the following system of governing equations along with initial and boundary conditions, which describe 3-D subsurface flow, 2-D overland flow, and 1-D channel flow. The governing equation for 3-D subsurface flow is basically the Richard's equation of which the derivation can be found elsewhere [Yeh et al., 1995].

Governing Equation for 3-D Subsurface Flow

$$F \frac{\partial h}{\partial t} = \nabla \cdot [\mathbf{K} \cdot (\nabla h + \nabla z)] + q; \quad F = \alpha' \frac{\theta}{n_e} + \beta' \theta + n_e \frac{dS}{dh} \quad (2.1)$$

where h is the pressure head; t is time; α' and β' are the modified compressibilities of the soil matrix and liquid fluid, respectively, n_e is the effective porosity, θ is the moisture content; S is the saturation; \mathbf{K} is the hydraulic conductivity tensor; z is the potential head, q is the source and/or sink. The hydraulic conductivity \mathbf{K} is given by

$$\mathbf{K} = \frac{\rho g}{\mu} \mathbf{k} = \frac{\rho g}{\mu} \mathbf{k}_s \mathbf{k}_r = \mathbf{K}_{so} \mathbf{k}_r \quad (2.2)$$

where ρ is the fluid density; μ is the fluid dynamic viscosity; \mathbf{k} is the permeability tensor; \mathbf{k}_s is the saturated permeability tensor; \mathbf{k}_r is the relative permeability or relative hydraulic conductivity; \mathbf{K}_{so} is the referenced saturated hydraulic conductivity tensor. The Darcy velocity is calculated as follows

$$\mathbf{V} = -\mathbf{K} \cdot (\nabla h + \nabla z) \quad (2.3)$$

Initial Conditions for 3-D Subsurface Flow

$$h = h_i(x, y, z) \quad \text{in } R, \quad (2.4)$$

where R is the region of interest and h_i is the prescribed initial condition, which can be obtained by either field measurements or by solving the steady state version of Eq. (2.1).

Boundary Conditions for 3-D Subsurface Flow

Dirichlet Conditions:

$$h = h_d(x_b, y_b, z_b, t) \quad \text{on } B_d \quad (2.5)$$

Neumann Conditions:

$$-\mathbf{n} \cdot \mathbf{K} \cdot \nabla h = q_n(x_b, y_b, z_b, t) \quad \text{on } B_n, \quad (2.6)$$

Cauchy Conditions:

$$-\mathbf{n} \cdot \mathbf{K} \cdot (\nabla h + \nabla z) = q_c(x_b, y_b, z_b, t) \quad \text{on } B_c, \quad (2.7)$$

Variable Conditions - During Precipitation Period:

$$h = h_p(x_b, y_b, z_b, t) \quad \text{on } B_v \quad (2.8a)$$

or

$$-\mathbf{n} \cdot \mathbf{K} \cdot (\nabla h + \nabla z) = q_p(x_b, y_b, z_b, t) \quad \text{on } B_v, \quad (2.8b)$$

Variable Conditions - During Non-precipitation Period:

$$h = h_p(x_b, y_b, z_b, t) \quad \text{on } B_v, \quad (2.8c)$$

or

$$h = h_m(x_b, y_b, z_b, t) \quad \text{on } B_v, \quad (2.8d)$$

or

$$-\mathbf{n} \cdot \mathbf{K} \cdot (\nabla h + \nabla z) = q_e(x_b, y_b, z_b, t) \quad \text{on } B_v, \quad (2.8e)$$

River Conditions:

$$-\mathbf{n} \cdot \mathbf{K}_s \mathbf{K}_r \cdot (\nabla h + \nabla z) = \mathbf{K}_r \cdot \frac{H_R - (h + z)}{d_R} \quad \text{on } B_R, \quad (2.9)$$

where (x_b, y_b, z_b) is the spatial coordinate on the boundary; \mathbf{n} is an outward unit vector normal to the boundary; h_d , q_n , and q_c are the prescribed Dirichlet functional value, Neumann flux, and Cauchy flux, respectively; B_d , B_n , and B_c are the Dirichlet, Neumann, and Cauchy boundary, respectively; B_v is the Variable boundary and B_R is the River boundary; h is the allowed ponding depth and q is the throughfall of precipitation,

respectively, on the Variable boundary; h_m is the allowed minimum pressure on the Variable boundary; and q_e is the allowed maximum evaporation rate on the Variable boundary, which is the potential evaporation; K_R and d_R are the hydraulic conductivity and the thickness of the sediment overlying the River boundary, respectively; H_R is the water head on the top surface of the sediment. Only one of Eqs. (2.8a) through (2.8e) is used at any point on the Variable boundary at any time.

The governing equation for 1-D channel flow is derived based on the concept of water budget. The sources/sinks for each channel reach include (1) pumping into/from other channel reaches or a specific location in the subsurface as well as pumping into a retention pond, (2) precipitation/evaporation, (3) discharge into/from neighboring channel reaches or through upstream/downstream boundaries, and (4) interaction with the geologically-connected subsurface system. *A channel reach here is defined as a channel section with two connectors as its two ends. A connector can be a upstream boundary gate, an interior gate, a downstream boundary gate, a weir, a dead end, or a structure-free joint.* The above definition will be used throughout this document.

Governing Equation for 1-D Channel Flow

$$B(j) \frac{dHRT(j)}{dt} = QT(j) + QP(j) + QUD(j) + QS(j) \quad j \in [1, NORH] \quad (2.10)$$

where $B(j)$ is the top surface area of the j -th channel reach; $HRT(j)$ is the water stage of the j -th channel reach; $QT(j)$ is the net volumetric flow rate into the j -th channel reach through the water surface due to precipitation/evaporation; $QP(j)$ is the net volumetric flow rate to the j -th channel reach through pumping; $QUD(j)$ is the net volumetric flow rate from the neighboring channel reaches through control structures to the j -th channel reach; $QS(j)$ is the net volumetric flow rate from the subsurface through the channel-subsurface interface to the j -th channel reaches; $NORH$ is the number of channel reaches discretized for simulation.

$QT(j)$ is obtained by multiplying the precipitation rate by the top surface area of the j -th channel reach.

$QS(j)$ is obtained by multiplying the exfiltration (positive value) or infiltration rate (negative value) from the

subsurface media to the j -th channel by the wetted area of the channel-subsurface interface. The exfiltration or infiltration rate depends on both the pressure head in subsurface media and water depth in channel. It is modeled with an exchange term in which the storage effect of a thin layer between channel and subsurface media is neglected. Computations of $QS(j)$ via an iterative solution procedure between channel and subsurface media are detailed in Appendix B.

$QUD(j)$ can be decomposed into two parts: the upstream and downstream flow rates of the j -th channel reach, i.e., $QUD(j) = QUD^u(j)$ (the upstream flow rate) + $QUD^d(j)$ (the downstream flow rate).

For the case of upstream flow rate, the following two situations are considered.

- (1) Submerged structure: For a submerged structure, the flow rate is controlled by both the headwater and tailwater of the upstream gate of channel reach j . The incoming flow rate can be calculated by

$$QUD^u(j) = Q^{upstream}(HUP, HRT(j)) \quad (2.11a)$$

where HUP is the stage of the upstream reach of reach j and $Q^{upstream}(HUP, HRT(j))$ is the upstream flow rate that is a function of the stages of both reach j and its upstream reach. If the upstream gate is an interior gate, HUP is computed and users must provide a rating curve describing stage difference-dependent flow rate (Q vs. ΔH) for calculating $QUD^u(j)$. If the upstream gate is a global boundary node, users must provide two rating curves, t vs. HUP and Q vs. ΔH , for calculating $QUD^u(j)$.

- (2) Non-submerged structure: For a non-submerged structure, the flow rate is controlled only by the headwater of the upstream gate of channel reach j . The incoming flow rate can be calculated by

$$QUD^u(j) = Q^{upstream}(HUP) \quad (2.11b)$$

where $Q^{upstream}(HUP)$ is the upstream flow rate that is a function HUP only. If the upstream gate is an interior gate, HUP is computed and users must provide a rating curve describing water depth-dependent flow rate (Q vs. h) for calculating $QUD^u(j)$. If the upstream gate is a global boundary, then a time-dependent flow rate must be specified with a rating curve (Q vs. t) by users for computing

$QUD^u(j)$ simply because HUP may be a function of time.

Likewise, two situations are considered for the case of downstream flow rate as in the following.

- (1) Submerged structure: For a submerged structure, the flow rate is controlled by both the headwater and tailwater of the downstream gate of channel reach j. The incoming flow rate can be calculated by

$$QUD^d(j) = Q^{\text{downstream}}(HRT(j), HDOWN) \quad (2.12a)$$

where HDOWN is the stage of the downstream reach of reach j and $Q^{\text{downstream}}(HRT(j), HDOWN)$ is the downstream flow rate that is a function of the stages of both reach j and its downstream reach. If the downstream gate is an interior gate, HDOWN is computed and users must provide a rating curve describing stage difference-dependent flow rate (Q vs. ΔH) for calculating $QUD^d(j)$. If the downstream gate is a global boundary node, users must provide two rating curves, t vs. HDOWN and Q vs. ΔH , for calculating $QUD^d(j)$.

- (2) Non-submerged structure: For a non-submerged structure, the flow rate is controlled only by the headwater of the downstream gate of channel reach j. The incoming flow rate can be calculated by

$$QUD^d(j) = Q^{\text{downstream}}(HRT(j)) \quad (2.12b)$$

where $Q^{\text{downstream}}(HRT(j))$ is the downstream flow rate that is a function of HRT(j) only. Since HRT(j) is computed at the node, users need to provide a rating curve describing water depth-dependent flow rate (Q vs. H) to calculate $QUD^d(j)$ for both cases that the downstream gate is an interior gate and it is a global boundary.

To summarize the above, water depth-dependent flow rates are prescribed for the case of non-submerged gates, while stage difference-dependent flow rates are prescribed for the case of submerged gates. For the interior gate, the status may change between submerged and non-submerged. Thus, users need to provide two rating curves, Q vs. ΔH for the submerged case and Q vs. h for the non-submerged case, for each interior gate. If the gate is a global boundary, two cases are considered: upstream and downstream boundary

gates. For the case of upstream boundary gate, only non-submerged flow is taken into account and one rating curve of Q vs. t needs to be given. Like the interior gate, a downstream boundary gate can be submerged or non-submerged, depending on the stage its tailwater. Since the tailwater is not included in the simulated domain, its time-dependent stage profile must be provided by users. As a result, three rating curves need to be given for a downstream boundary gate: Q vs. ΔH for submerged flow, Q vs. h for non-submerged flow, and t vs. H for its outside tailwater.

If the control structure is a weir, two rating curves (one for the submerged case and the other for the non-submerged case) must be given to compute flow rates, just like those for interior gates. The only difference between interior gates and weirs is that interior gates can be controlled open or close whereas the openings of weirs don't change with time.

Initial Conditions for 1-D channel flow

$$HRT(j) = HRT(j)_i \quad j \in [1, NORH] \quad (2.13)$$

where $HRT(j)_i$ is the prescribed initial water stage of the j -th channel reach.

The 2-D overland module includes two systems: overland flow and retention ponds. The governing equation for retention ponds is derived based on the concept of water budget, while the governing equation for overland flow is a spatial-temporal nonlinear partial differential equation. The sources/sinks for each retention pond consist of (1) pumping from canal, (2) precipitation/evaporation, (3) seepage into/from surrounded dikes, (4) interaction with the geologically-connected subsurface system, and (5) flow out of the outlets of retention ponds.

Governing Equation for Retention Ponds

$$A(j) \frac{\partial H(j)}{\partial t} = QT(j) + QP(j) - QU(j) + QS(j) \quad j \in [1, N] \quad (2.14)$$

where $A(j)$ is the surface area of the j -th retention pond when the water surface elevation is $H(j)$; $H(j)$ is the water surface elevation of the j -th retention pond; $QT(j)$ is the net volumetric flow rate through the water

surface due to precipitation/evaporation to the j-th retention pond; $QP(j)$ is the net volumetric flow rate pumping wells to the j-th retention pond, or vice versa; $QU(j)$ is the net volumetric flow rate flowing out of the outlet of the j-th retention pond; $QS(j)$ is the net volumetric flow rate from the subsurface media/dike to the j-th retention pond, or vice versa; N is the number of retention pond in the region of interest. QU and QS of retention ponds are simulated similar to the QUD and QS of channel. $QS(j)$ depends on both water surface elevation of the retention pond and total head in subsurface media, while $QU(j)$ depends only on water surface elevation of the retention pond.

Initial Conditions for Retention Ponds

$$H(j)=H(j)_i \quad j \in [1,N] \quad (2.15)$$

where $H(j)_i$ is the prescribed initial water surface elevation of the j-th retention pond.

Governing Equation for 2-D Overland Flow

The continuity equation is represented as

$$\frac{\partial h}{\partial t} + \nabla \cdot (\mathbf{V}h) = r - i + q \quad (2.16)$$

where h is water depth; \mathbf{V} is a velocity vector; q is the flux rate from retention ponds, r is rainfall rate, and i is infiltration rate. The infiltration rate i is obtained by an iterative solution between 2-D overland flow and 3-D subsurface flow, so is the flux rate q between 2-D retention pond water budget and 3-D subsurface flow. Detail numerical implementation of this iterative procedure will be given in Appendix B. The velocity vector is calculated by the following equation (2.17).

$$\mathbf{V} = -\frac{a}{n} \left[\frac{h}{1 + (\nabla z_0)^2} \right]^{\frac{2}{3}} \frac{\nabla(h + z_0)}{\sqrt{|\nabla(h + z_0)|}} \quad (2.17)$$

where n is Manning coefficient, a is a coefficient with the dimension $[L^{1/3}/T]$, and z_0 is ground-surface elevation. The typical value of a is $1 \text{ m}^{1/3}/\text{s}$. Substituting Eq. (2.17) into Eq. (2.16), yields the continuity equation:

$$\frac{\partial h}{\partial t} = \nabla \cdot [K \nabla (h + z_0)] + r - i + q \quad (2.18)$$

in which

$$K = \frac{a}{n} h^{5/3} \frac{1}{(1 + (\nabla z_0)^2)^{2/3}} \cdot \frac{1}{\sqrt{|\nabla (h + z_0)|}} \quad (2.19)$$

Initial Conditions for 2-D Overland Flow

$$h = h_i(x, y) \quad \text{in } R, \quad (2.20)$$

where R is the region of interest and h_i is the prescribed initial condition.

Boundary Conditions for 2-D Overland Flow

Dirichlet Conditions: a prescribed head (water depth)

$$h = h_d(x_b, y_b, t) \quad \text{on } B_d \quad (2.21)$$

Cauchy Conditions: a prescribed flux rate

$$-\mathbf{n} \cdot K(\nabla h + \nabla z_0) = q_c(x_b, y_b, t) \quad \text{on } B_c, \quad (2.22)$$

Radiation Conditions: a prescribed rating curve h vs. q_n

$$-\mathbf{n} \cdot K(\nabla h + \nabla z_0) = q_n(h(x_b, y_b, t)) \quad \text{on } B_n, \quad (2.23)$$

where (x_b, y_b) is the spatial coordinate on the boundary; \mathbf{n} is an outward unit vector normal to the boundary; h_d , q_c , and q_n are the prescribed Dirichlet functional value, Cauchy flux, and the flux obtained from rating curve h vs. q_n , respectively; B_d , B_c , and B_n are the Dirichlet, Cauchy, and Radiation boundary, respectively.

2.2 Description of COSFLOW subroutines

COSFLOW consists of a MAIN program and 119 subroutines required to perform simulation of the coupled flow system. The program structure of COSFLOW is illustrated in Figure 2.1 through Figure 2.5. The functions of the MAIN program and the subroutines are described below. To focus on the flow modules, we only describe subroutines that are associated with flow modules.

Program MAIN

The MAIN is used to specify the sizes of all arrays and control the entire sequence of operations. The procedure of reading data input for the model is also anchored by the MAIN. The subroutines FILES, RINPUT1, LINEAR, RINIT, MVINIT, RPINP2, RINPUT2, WRITIM, CHTIM2, DATA1D, DATA2D, DATA3D, and RPDATA are called to read and prepare all the information related to 1-D, 2-D, 3-D, and retention pond simulations, respectively. The other subroutines are called for the simulation of 1-D canal, 2-D overland, 3-D subsurface, and retention pond flow problems. All the subroutines are described in details as follows.

1 Subroutine FILES

The subroutine FILES is called by the program MAIN to read all the filenames via a super file. Those files are then open for the requisition of this simulation. Subroutines GEOM is called to read 3-D geometry data, i.e., nodal coordinates and element indices, RINPUT is to read the rest 3-D input data, e.g., option parameter, soil properties, etc., GEOM2 is to read geometry data of 2-D overland region.

2 Subroutine STRIP

This subroutine is called by subroutine FILES to strip the filenames provided by the super file.

3 Subroutine GEOM

This subroutine is called by subroutine FILES to read the geometry data of 3-D subsurface region. The coordinate data are ahead of GN card, whereas the element indices are ahead of GE4, GE6, or GE8 card.

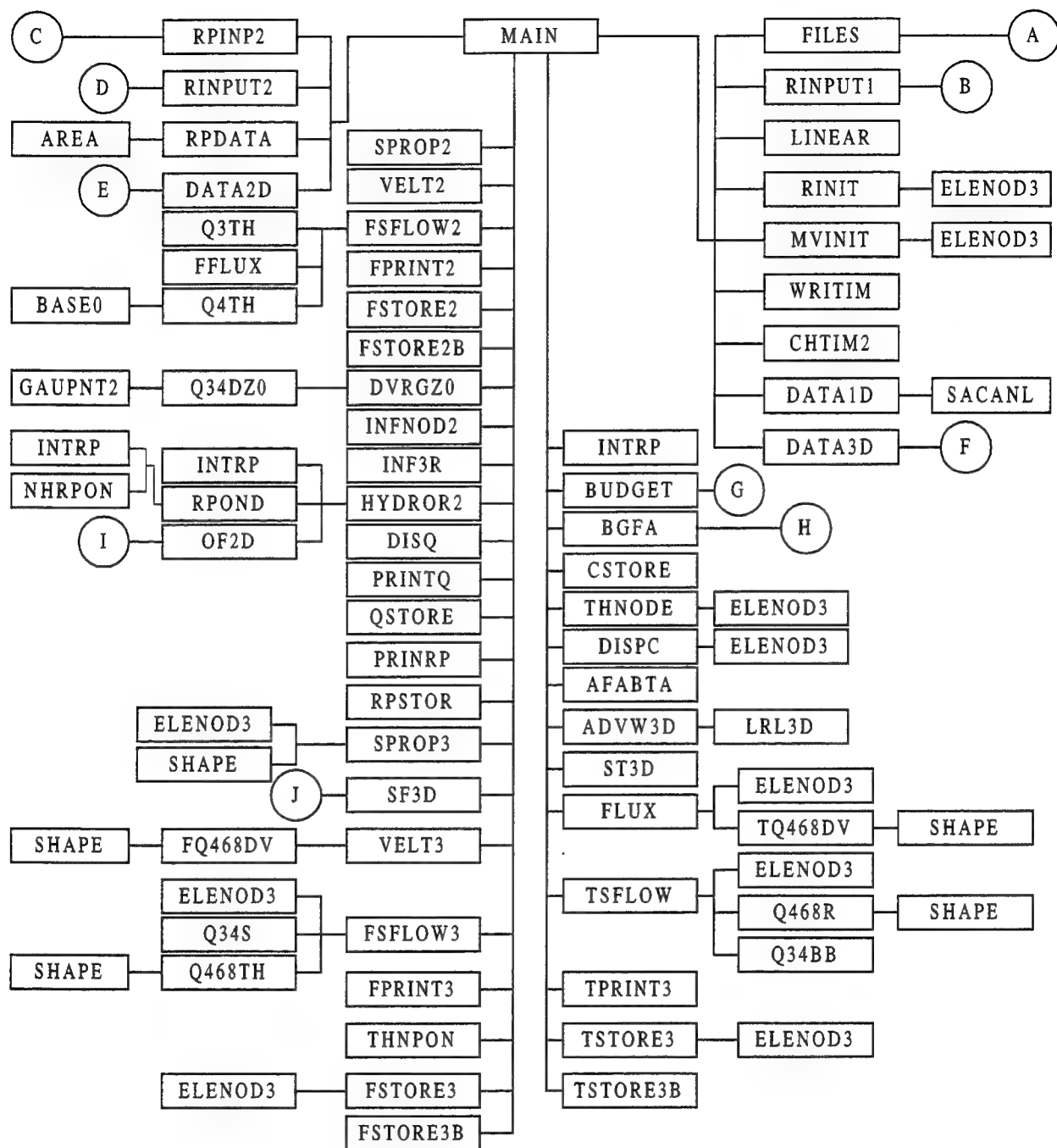


Figure 2.1 Program structure of COSFLOW

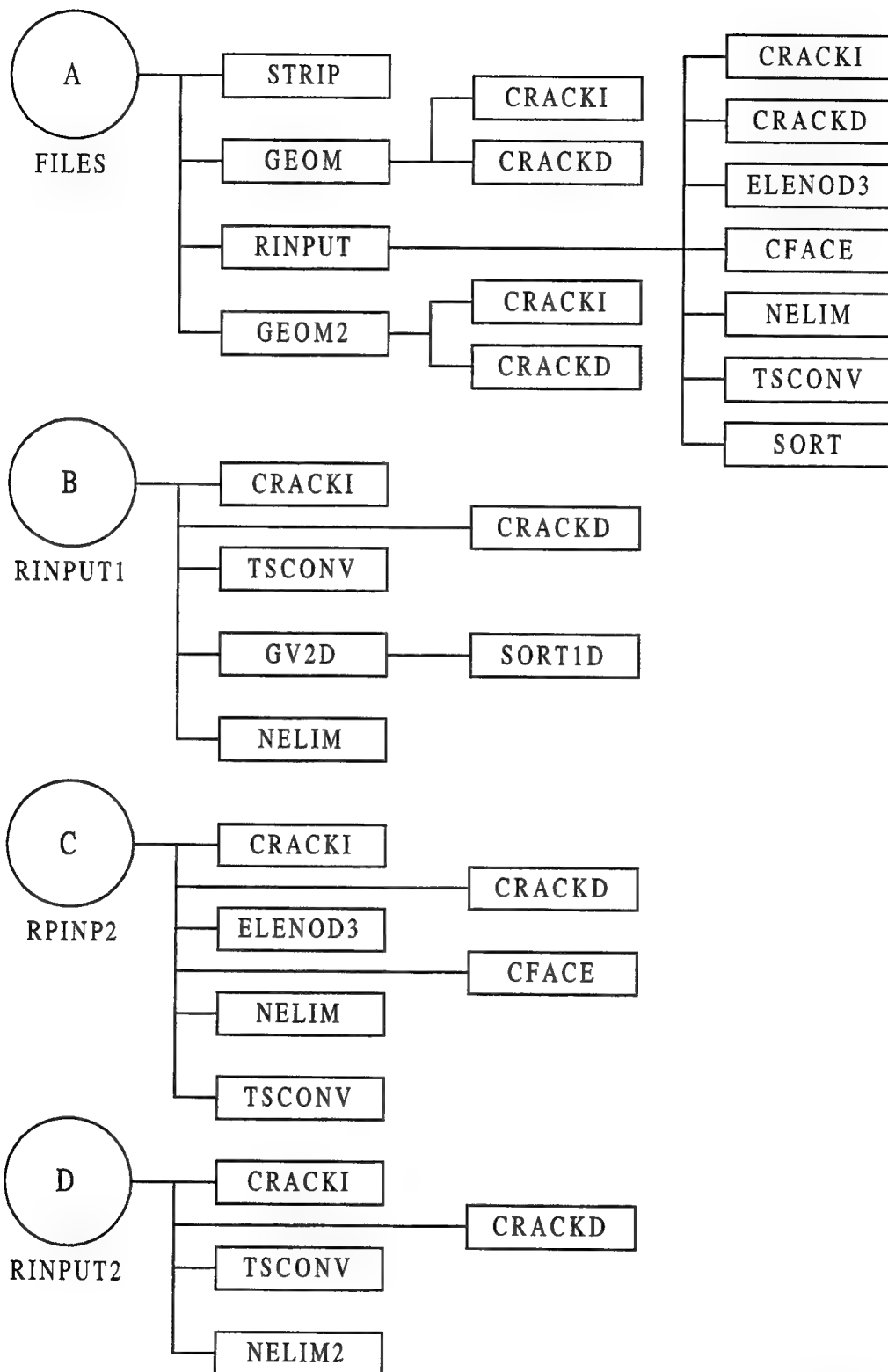


Figure 2.2 Program structure of subroutines FILES, RINPUT1, RPINP2, RINPUT2

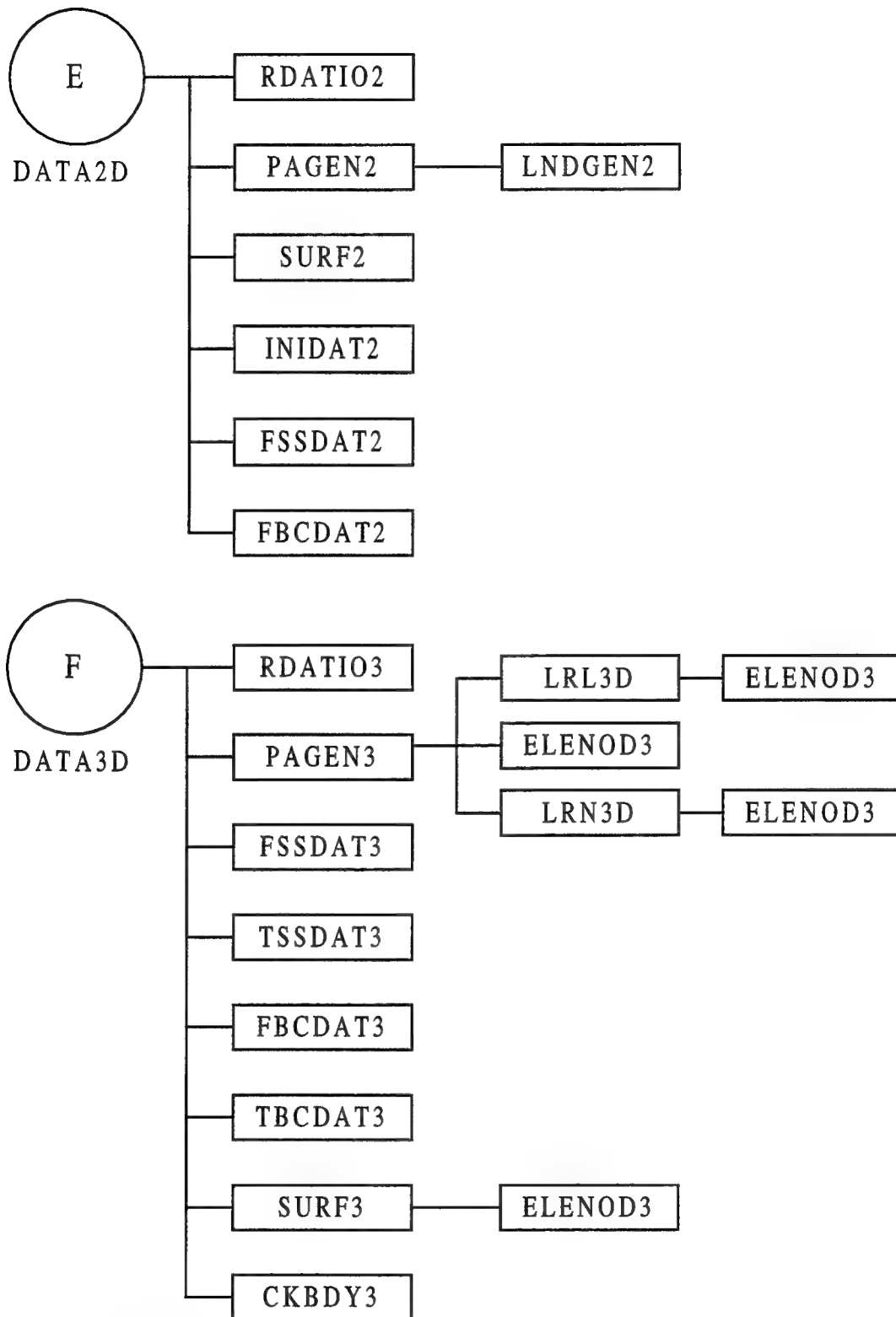


Figure 2.3 Program structure of subroutines DATA2D and DATA3D

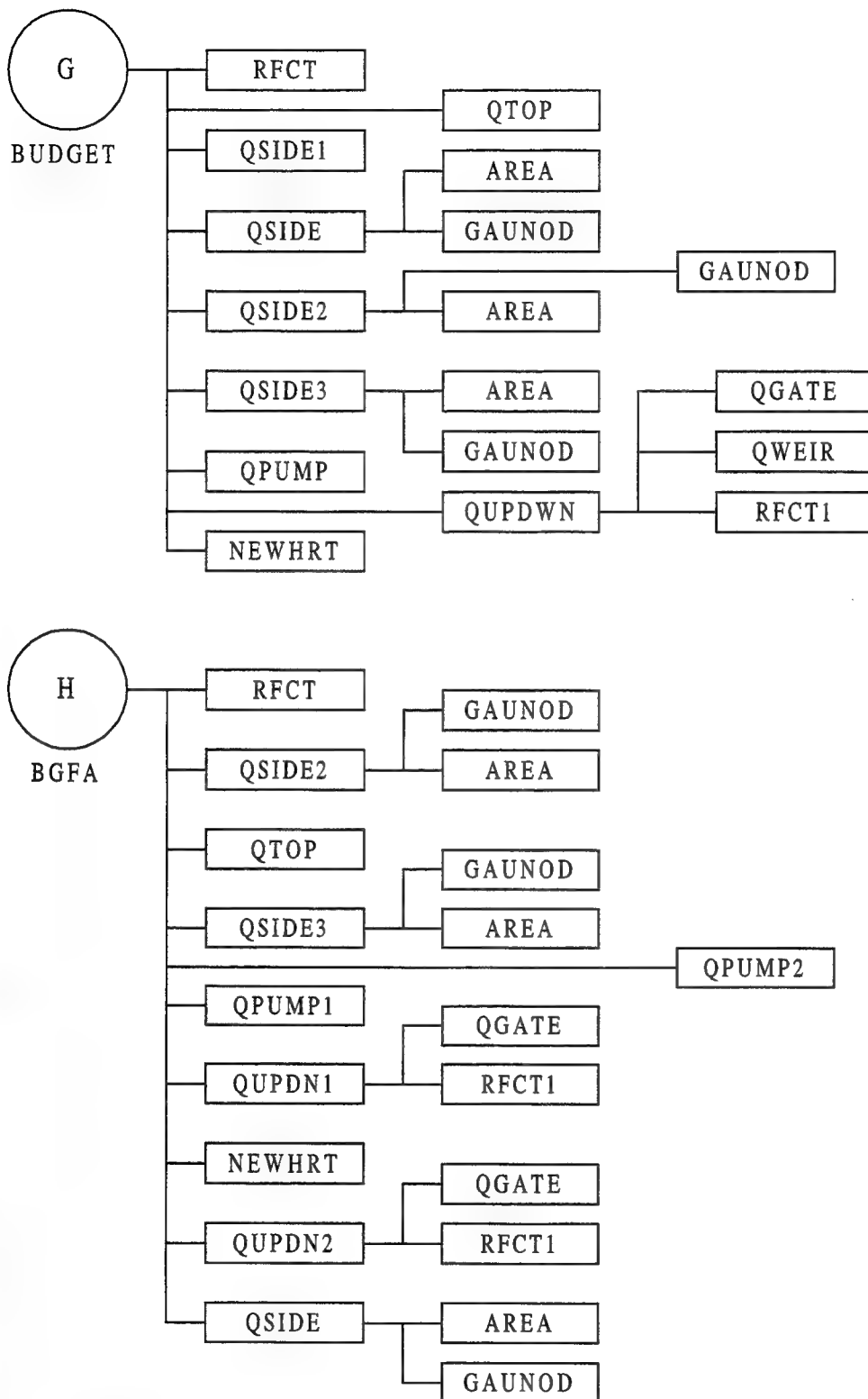


Figure 2.4 Program structure of subroutines BUDGET and BGFA

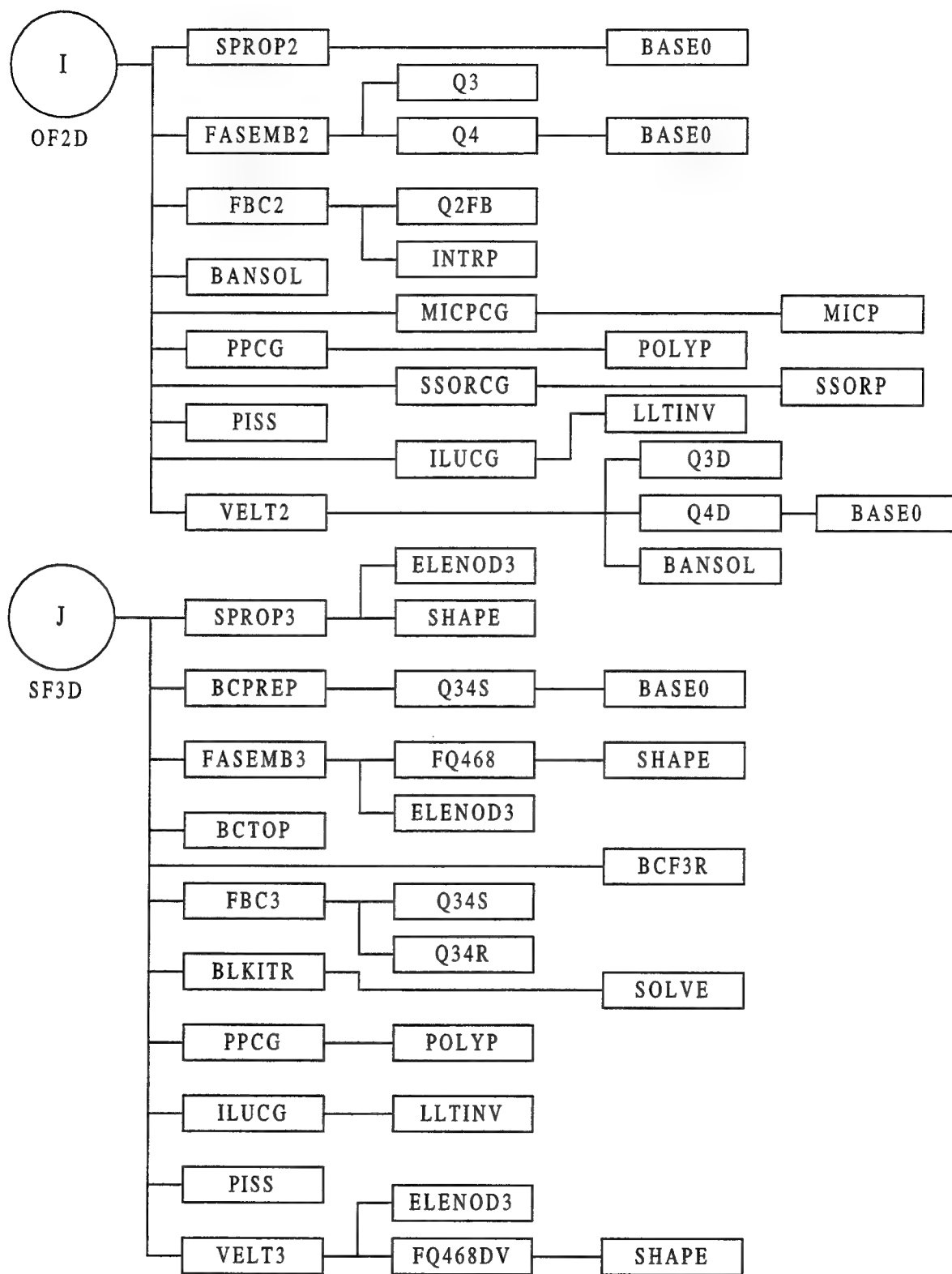


Figure 2.5 Program structure of subroutines OF2D and SF3D

Users can refer Appendix A for more detailed information.

4 Subroutine RINPUT

This subroutine is called by subroutine FILES to read all the subsurface data excluding geometry information. The contents to be read consist of analysis options, material properties, boundary conditions, and initial condition. Subroutines CRACKI, CRACKD, ELENOD3, CFACE, NELIM, TSCONV, and SORT are called here to prepare all the data required for the simulation.

5 Subroutine GEOM2

This subroutine is called by subroutine FILES to read the geometry data of 2-D overland region. The coordinate data are ahead of ND card, whereas the element indices are ahead of either E4Q or E3T card. Users can refer Appendix A for more detailed information.

6 Subroutine RINPUT1

This subrouitne is called by the program MAIN to read the information required for the simulation of 1-D canal flow. The information includes the specification of groundwater nodes making up the canal reach, geometric attributes of a canal reach, water stage attributes of a canal reach, control structures at the reach node, pumping station at a reach node, siphon station at a reach node, canal junctions, precipitation data, option parameters, and iteration parameters, etc..

7 Subroutine RINPUT2

This subroutine is called by the program MAIN to read the required information excluding geometry data for the simulation of 2-D overland flow. The information includes option parameters, iteration parameters, time control parameters, output control parameters, cross section parameters for hydrograph, material property parameters, rainfall information, depth boundary condition, flux boundary condition, radiation boundary condition, and initial boundary condition, etc..

8 Subroutine RPINP2

This subroutine is called by the program MAIN to read the required information for the simulation of retention ponds. The information includes the initial condition, geometry of retention ponds, rainfall data, outlet information, and intra-boundary data, etc..

9 Subroutine CRACKI

This subroutine is called by subroutines GEOM, RINPUT, GEOM2, RINPUT1, RPINP2, and RINPUT2 to count the number of integer data included in a record which is composed of characters, real and integer numbers separated by a blank or a comma. The record is then packed to the buffer. The last, the integers with correct format are read out of the buffer.

10 Subroutine CRACKD

This subroutine is called by subroutines GEOM, RINPUT, GEOM2, RINPUT1, RPINP2, and RINPUT2 to count the number of real data included in a record which is composed of characters, reals and integers separated by a blank or a comma. The record is then packed to the buffer. The last, the reals with correct format are read out of the buffer.

11 Subroutine CFACE

This subroutine is called by subroutines RINPUT and RPINP2 to transform the element face ID generated from GMS to the convention used in the computation kernel.

12 Subroutine NELIM

This subroutine is called by subroutines RINPUT, RINPUT1, and RPINP2 to prepare an array consisting of nodal numbers from a set of element face data.

13 Subroutine NELIM2

This subroutine is called by subroutine RPINP2 to prepare an array consisting of nodal numbers from a set of line segment data.

14 Subroutine TSCONV

This subroutine is called by subroutines RINPUT, RINPUT1, RPINP2, and RINPUT2 to group the specified xy series into a set of time series or rating curves for specific purpose.

15 Subroutine SORT

The subroutine is called by subroutine RINPUT to compress the material ID so that there is no skip number in the material IDs. Hence, the material ID is expected to increase sequentially from 1 to the number of materials.

16 Subroutine GV2D

This subroutine is called by subroutine RINPUT1 to generate 2-D vertical interface mesh between canal and subsurface. This subroutine is called when IFACE is equal to 2, which is specified in IFS card of 1-D canal model file (see Appendix A.).

17 Subroutine SORT1D

This subroutine is called by subroutine GV2D to find the maximum elevation among the 3-D subsurface nodes on the bottom of a reach.

18 Subroutine RINIT

This subroutine is called by MAIN to read spatial varied initial condition (I.C.) of 3-D subsurface regime. The format of the I.C. file can be either an ASCII text file or a binary file. The pressure head at every node must be provided in this I.C. file.

19 Subroutine WRITIM

This subroutine is called by the program MAIN to rearrange the index for the selected printing and saving information of each groundwater time step.

20 Subroutine CHTIM2

This subroutine is called by the program MAIN to rearrange the index for the selected printing and saving information of each overland time step.

21 Subroutine DATA3D

The subroutine DATA3D is called by the program MAIN to prepare and write out all the information related to 3-D subsurface flow simulation. It calls subroutines RDATIO3 to write material properties, soil property functions, and geometry data for the 3-D region of interest. Subroutines PAGEN3 and SURF3 are called to generate pointer and boundary arrays. It also calls subroutines FSSDAT3, TSSDAT3, FBCDAT3, and TBCDAT3 to write out sources/sinks as well as boundary conditions for flow and transport simulations, respectively.

22 Subroutine DATA2D

The subroutine DATA2D is called by the program MAIN to prepare and write out all the information related to 2-D overland flow simulation. It calls subroutines RDATIO2 to write out material properties and geometry data for the 2-D region of interest. Subroutines PAGEN2 and SURF2 are called to generate pointer and boundary arrays. Subroutine INIDAT2 is called to read initial conditions. It also calls subroutines FSSDAT2 and FBCDAT2 to write out sources/sinks as well as boundary conditions, respectively..

23 Subroutine DATA1D

This subroutine prints out input data related to 1-D channel simulation. The data include channel reach information, 1-D channel element information, precipitation information for channel reaches, time step sizes for 1-D channel simulation, the initial condition for channel reaches, and the information of channel boundary sides. It also calls subroutine SACANL to calculate the coefficients involved to compute the areas of channel reach top and side/bottom surfaces for later use.

24 Subroutine RPDATA

This subroutine writes out the information for the simulation of retention ponds. The information includes the location of retention ponds, the size of each pond, the description of intra-boundaries, the outlets on retention ponds, rainfall, and initial conditions. It also calls subroutine AREA to compute an elemental area

on the interface between the retention pond and subsurface for the estimation of bottom area of each retention pond.

25 Subroutine RDATIO3

The subroutine RDATIO3 is called by subroutine DATA3D to write out the material properties, soil property functions, and geometric data for the area of interest.

26 Subroutine PAGEN3

This subroutine is called by subroutine DATA3D to preprocess pointer arrays that are needed to store the global matrix in compressed form and to construct the subregional block matrices. The pointer arrays automatically generated in this subroutine include the global node connectivity (stencil) GNOJCN(J,N), regional node connectivity LNOJCN(J,I,K), total node number for each subregion NTNPLR(K), bandwidth indicator for each subregion LMAXDF(K), and partial fill-up for the mapping array between global node number and local subregion node number GNPLR(I,K) with $I = \text{NNPLR}(K) + 1$ to $\text{NTNPLR}(K)$. Here GNOJCN(J,N) is the global node number of J-th node connected to the global node N; LNOJCN(J,I,K) is the local node number of the J-th node connected to the local node I in K-th subregion; NTNPLR(K) is the total number of nodes in the K-th subregion, including the interior nodes, the global boundary nodes, and intra-boundary nodes; LMAXDF(K) is the maximum difference between any two nodes of any element in K-th subregion; and GNPLR(I,K) is the global node number of I-th local-region node in the K-th subregion. These pointer arrays are generated based on the element connectivity IE(M,J), the number of node for each subregion NNPLR(K), and the mapping between global node and local-region node GNLR(I,K) with $I = 1, \text{NNPLR}(K)$. Here IE(M,J) is the global node number of J-th node of element M; NNPLR(K) is the number of nodes in the K-th subregion including the interior nodes and the global boundary nodes but not the intra boundary nodes. This subroutine also re-arranges LRN3(J,N) such that $\text{LRN3}(1,N) = N$.

27 Subroutine LRL3D

This subroutine is called by subroutine PAGEN3 to prepare the arrays regarding the number of global elements and the associated global element number connected to each global node.

28 Subroutine LRN3D

This subroutine is called by subroutine PAGEN3 to prepare the arrays regarding the number of global nodes and the associated global node number connected to each global node.

29 Subroutine SURF3

Subroutine SURF3 identifies the boundary sides, sequences the boundary nodes, and computes the directional cosine of the surface sides. The mappings from boundary nodes to global nodes are stored in NPBB3(I) (where NPBB3(I) is the global node number of the I-th boundary node). The boundary node numbers of the four nodes for each boundary side are stored in ISB3(I,J) (where ISB3(I,J) is the boundary node number of I-th node of J-th side, I = 1 to 4). There are six, five, or six sides for each element. Which of these sides is the boundary side is determined automatically in the subroutine SURF3 and is stored in ISB3(5,J). The global element number, to which the J-th boundary side belongs, is also pre-processed in the subroutine SURF and is stored in ISB3(6,J). The directional cosines of the J-th boundary side are computed and stored in DCOSB3(I,J) (where DCOSB3(I,J) is the directional cosine of the J-th surface with I-th coordinate, I = 1 to 3). The information contained in NPBB3, ISB3, and DCOSB3, along with the number of boundary nodes and the number of boundary sides is returned to subroutine DATA3D for other users.

30 Subroutine ELENOD3

This subroutine returns three integers indicating the number of nodes, the number of element sides, and the shape of the specific element, respectively.

31 Subroutine FSSDAT3

The subroutine FSSDAT3 is called by subroutine DATA3D to write out the sources/sinks profiles, nodes, and/or elements for flow simulations. The source/sink type for each node/element has been prepared

previously so that the information can be printed out.

32 Subroutine FBCDAT3

The subroutine FBCDAT3 writes out all the boundary condition information including the specification of the global node/element numbers of the boundary, and assignment of the boundary profile type to each node/element.

33 Subroutine RDATIO2

Subroutine RDATIO2 prints material and domain data including material properties, the global node information, and element incidences for the simulation of 2-D overland flow.

34 Subroutine PAGEN2

This subroutine called subroutine LNDGEN2 to generate pointer arrays including the number of elements and associated global element number as well as the number of global nodes and the associated global element number connecting to each global node in 2-D overland flow domain. This subroutine also rearranges $LRN2(J,N)$ such that $LRN2(1,N) = N$.

35 Subroutine LNDGEN2

This subroutine is called by the controlling subroutine PAGEN2 to preprocess the two pointer arrays for the simulation of 2-D overland flow. One is the global node connectivity (stencil) $LRN2(J,N)$, where $LRN2(J,N)$ is the global node number of J-th node connected to the global node N. This pointer array serves the purposes of assembling the global matrix in compressed form when pointwise solution methods are used. The other pointer array is the global elements connected to nodes $LRL2(K,N)$, where $LRL2(K,N)$ is the global element number of the K-th elements connected to the global node N. These two pointer arrays are generated based on the element connectivity $IE2(M,J)$. Here $IE2(M,J)$ is the global node number of J-th node of element M.

36 Subroutine SURF2

Subroutine SURF2 identifies the boundary sides, sequences the boundary nodes, and computes the directional cosine of the surface sides. The mapping from boundary nodes to global nodes are stored in NPBB2(I) (where NPBB2(I) is the global node number of I-th boundary node). The element number associated with the boundary sides are stored in MBES2. The length and directional cosines for each side are stored in DLB and DCOSXB and DCOSZB, respectively. The local and global nodal number of two nodes of each side are stored in ISB2. The information containing in NPBB2, MBES, ISB2, DCOSXB, and DCOSZB along with the number of boundary nodes and the number of boundary sides are returned to subroutine DATA2D for other users.

37 Subroutine INIDAT2

This subroutine is called by subroutine DATA2D to writes out the initial condition, i.e., the water depth at each 2-D overland node, for the simulation of 2-D overland flow.

38 Subroutine FSSDAT2

Subroutine FSSDAT2 prints the element source/sink as well as the point source/sink data including source/sink profiles as functions of time, the source/sink nodes, the source/sink sides, and profiles types assigned to each source/sink element or source/sink node.

39 Subroutine FBCDAT2

Subroutine FBCDAT2 prints the boundary conditions including Dirichlet, Cauchy, and Radiation conditions. For each type of boundary conditions, information includes the boundary-value profiles as functions of times, boundary nodes, boundary sides, and boundary-value profiles assigned to each of all boundary sides or boundary nodes.

40 Subroutine SACANL

This subroutine calculates the coefficients involved to compute the areas of channel reach top and side/bottom surfaces for later use. Both the top and side/bottom surface areas can be written as linear functions

of the water stage of channel reaches. For reach i, the following equations can be derived.

$$A^S(i) = \sum_{j=1}^{ncer(i)} L(i,j) * [BW(i,j) + (HRT(i) - BE(i,j)) * (\sqrt{1 + s1(i,j)^2} + \sqrt{1 + s2(i,j)^2})] \quad (2.24)$$

The above two equations can be rewritten as

$$A^T(i) = a(i) + b(i) * HRT(i) \quad (2.25)$$

$$A^S(i) = c(i) + d(i) * HRT(i) \quad (2.26)$$

where $A^T(i)$ and $A^S(i)$ are the top and side/bottom surface areas of channel reach i, respectively; $ncer(i)$ is the number of channel elements included in channel reach i; $L(i,j)$ is the length of the j-th 1-D channel element in channel reach i; $BW(i,j)$ and $BE(i,j)$ are the bottom width and the bottom elevation, respectively, of the j-th 1-D channel element in channel reach i; $HRT(i)$ is the water stage of channel reach i; $s1(i,j)$ and $s2(i,j)$ are the right and left side slopes, respectively, of the j-th channel element in channel reach i; $a(i)$ and $b(i)$ are the zero-th and first order coefficients for the top surface area of reach i; $c(i)$ and $d(i)$ are the zero-th and first order coefficients for the side/bottom surface area of reach i. It is called by subroutine DATA1D.

41 Subroutine INTRP

This subroutine is called by the program MAIN, subroutines HYDROR2 and RPOND to compute the interpolated value on the provided profile. It uses the linear interpolation of the tabular data.

42 Subroutine BUDGET

This subroutine performs the 1-D channel simulation, with the consideration of water budget, through one subsurface time step (many channel time steps are thus included). For each channel time step, it calls subroutines RFCT, QSIDE1, QSIDE2, QSIDE3, QTOP, QSIDE, QPUMP, and QUPDOWN to obtain the incoming and outgoing fluxes for each reach. With the net flux and the given channel time step size, it calls subroutine NEWHRT to calculate the water stage of reaches at the new time being considered. This subroutine can automatically reduce the channel time step size based on the desired control procedure. In addition, it

makes up the deficit of the channel time step size by accomplishing extra channel time step(s). It is called by program MAIN when IFA = 0 is specified in ANG card of 1-D canal model file (see Appendix A).

43 Subroutine BGFA

This subroutine is specifically designed for the cases of South Florida to perform the 1-D channel simulation, with the consideration of water budget, through one subsurface time step (many channel time steps are thus included). For each channel time step, it calls subroutines RFCT, QTOP, QSIDE2, QSIDE3, QSIDE, QPUMP1, QPUMP2, QUPDN1 and QUPDN2 to obtain the incoming and outgoing fluxes for each reach. With the net flux and the given channel time step size, it calls subroutine NEWHRT to calculate the water stage of reaches at the new time being considered. This subroutine can automatically reduce the channel time step size based on the desired control procedure. In addition, it makes up the deficit of the channel time step size by accomplishing extra channel time step(s). It is called by program MAIN when IFA = 1 is specified in ANG card of 1-D canal model file (see Appendix A).

44 Subroutine RFCT

This subroutine finds the profile value at time T either by tabular interpolation or by analytical expression. For the latter case, the user-provided formula must be described.

45 Subroutine QTOP

This subroutine computes the flow rates to channel reaches due to precipitation/evapotranspiration. It is called by subroutines BUDGET and BGFA.

46 Subroutine QSIDE3

This subroutine computes the flow rates through the side/bottom surface of channel reaches from or into the subsurface. This approach calculates the flow rate through the composite planes that describe the real subsurface-channel interface. The user needs to specify channel boundary sides in IFS card of 1-D canal model file (see Appendix A). The computed flow rate is evenly distributed to the subsurface nodes associated

with this side for serving as point sources/sinks for the subsurface simulation. The flow rate through each side, which is specified according to the true geometry of the interface, is computed and distributed to the nodes of that side. One can read APPENDIX B.2 for detailed computation. It is called by subroutines BUDGET and BGFA.

47 Subroutine QSIDE2

This subroutine computes the flow rates through the side/bottom surface of channel reaches from or into the subsurface. This approach treats the flow from/into the subsurface as vertical plane sources/sinks and can be employed if the significance in the vertical direction (z-direction) needs to be emphasized while the channel width is still negligible when compared to the horizontal dimension of the simulated subsurface domain. The channel boundary sides (or plane elements) on the vertical plane are specified in IFS card of canal model file (see Appendix A). The flow rate through each side is computed here. The computed flow rate needs to be further multiplied by a factor of 2 for simulating the flow through the two side boundaries of a channel in practice. The multiplied flow rate is then evenly distributed to the subsurface nodes associated with this side for serving as point sources/sinks for the subsurface simulation. The flow rate through each simplified vertical plane is computed and distributed to the nodes of that side. One can read APPENDIX B.2 for detailed computation. It is called by subroutine BUDGET and subroutine BGFA.

48 Subroutine QSIDE1

This subroutine computes the flow rates through the side/bottom surface of channel reaches from or into the subsurface. This approach treats the subsurface contributions as line sources/sinks and can be used if both the width and the depth of channel are negligible when compared to the dimension of the simulated subsurface domain in all directions. The computed flow rate is evenly distributed to the subsurface nodes associated with this line for serving as point sources/sinks for the subsurface simulation. One can read APPENDIX B.2 for detailed computation. It is called by subroutine BUDGET.

49 Subroutine QSIDE

This subroutine computes the flow rates through the side/bottom surface of channel reaches from or into the subsurface based on the current canal's water surface elevation. The methodology is the same as that in subroutine QSIDE1, QSIDE2, or QSIDE3 according to the specification of IFS card in canal model file. This subroutine is called by subroutines BUDGET and BGFA.

50 Subroutine AREA

This subroutine computes the area enclosed by three or four points. The area is returned to subroutine QSIDE2, QSIDE3, or RPDATA

51 Subroutine OPUMP

This subroutine determines the controlled pumping flow rates to channel reaches. Water can be pumped from one reach to another reach, to a subsurface node, or to a retention pond. For the first case, the associated reach that water would be pumped from or into needs to be specified in CNP card of 1-D canal model file (see Appendix A). For the second or third cases, the associated subsurface node or the retention pond, respectively, needs to be located. A positive pumping flow rate to a reach implies a rise of water stage of the reach. Consequently, it will cause a drop of water level in the associated reach or a sink at the associated subsurface node/retention pond. It is called by subroutine BUDGET.

52 Subroutine QUPDWN

This subroutine computes the flow rate from one channel reach to its neighboring reaches, and vice versa. It calls subroutines RFCT1, QGATE and QWEIR to achieve the purpose. It is called by subroutine BUDGET. The following rules are employed in this subroutine to determine whether or not the gate should be opened. First, only is water flow allowed from the upstream reach to the downstream reach. Second, gates will be opened when either the headwater stage is higher than the prescribed upper stage limit or the tailwater stage is lower than the prescribed lower stage limit.

53 Subroutine QGATE

This subroutine computes the flow rate through controlled gates. Both modular and submerged flows are considered. Either rating curves (when $IRATEG = 1$) or analytical formulae (when $IRATEG = 0$) can be used to calculate discharge rate through gates. If $IRATEG = 0$ is specified, one needs to provide formulae and modify this subroutine. It is called by subroutines QUPDOWN, QUPDN1, and QUPDN2.

54 Subroutine QWEIR

This subroutine computes the flow rates through uncontrolled weirs. Both modular and submerged flows are considered. Either rating curves (when $IRATEW = 1$) or analytical formulae (when $IRATEW = 0$) can be used to calculate discharge rate through weirs. If $IRATEW = 0$ is specified, one needs to provide formulae and modify this subroutine. It is called by subroutine QUPDOWN.

55 Subroutine RFCT1

This subroutine finds the profile value at time T either by tabular interpolation or by analytical expression. For the latter case, the user-provided formula must be described.

56 Subroutine NEWHRT

This subroutine computes the new water stage of every channel reach based on the computed net flux and the given time step size. It is called by subroutines BUDGET and BGFA.

57 Subroutine QPUMP1

This subroutine functions similarly to subroutine QPUMP, but it is specifically designed for the case of South Florida. It is called by subroutine BGFA.

58 Subroutine QPUMP2

This subroutine functions similarly to subroutine QPUMP, but it is specifically designed for the case of South Florida and controlled by head series, i.e., $IFA = 2$ in ANG card of 1-D canal model file (see Appendix A). It is called by subroutine BGFA.

59 Subroutine QUPDN1

This subroutine achieves the same purpose as subroutine QUPDOWN. Since it is specifically designed for the case of South Florida, subroutines RFCT1 and QGATE are called here to compute the volumetric flow rates through gates. It is called by subroutine BGFA.

60 Subroutine QUPDN2

This subroutine achieves the same purpose as subroutine QUPDOWN. Since it is specifically designed for the case of South Florida and controlled by head series, i.e. IFA =2 in ANG card of 1-D canal model file (see Appendix A), only subroutines RFCT1 and QGATE are called here to compute the volumetric flow rates through gates. It is called by subroutine BGFA.

61 Subroutine GAUNOD

This subroutine is called by subroutines QSIDE1, QSIDE2, QSIDE3, and QSIDE to compute the base functions at quadrature points.

62 Subroutine SPROP2

This subroutine calculates the values of K at the Gaussian points in each element for 2-D overland flow simulation. The value K is represented as Eq. (2.19).

63 Subroutine VELT2

This subroutine calls Q4D or Q3D to evaluate the element matrices and the derivatives of the water surface elevation in Eq. (2.17). It then sums over all element matrices to form a matrix equation governing the velocity components at all nodal points. Subsequently, it calls Subroutine BANSOL to yield the solution.

To save computational time, an option of lumping the matrix is included. The velocity components can thus be solved point by point. The computed velocity field is then returned to program MAIN or subroutine OF2D through the argument.

64 Subroutine Q4D and Q3D

Subroutine Q4D and Q3D are called by the subroutine VELT2 to compute the element matrices given by:

$$QB(I,J) = \int_{R_e} N_i^e N_j^e dR \quad (2.27)$$

where N_i^e and N_j^e are the basis functions for nodal point i and j of element e , respectively. Subroutines Q4D and Q3D also evaluate the element load vector:

$$QRX(I) = - \int_{R_e} N_i^e K_1 i \cdot \nabla N_j^e H_j dR \quad (2.28)$$

$$QRY(I) = - \int_{R_e} N_i^e K_1 j \cdot \nabla N_j^e H_j dR \quad (2.29)$$

where

- H_j = the water surface elevation at nodal point j ;
- i = the unit vector along the x-coordinate,
- j = the unit vector along the y-coordinate,
- K_1 = K / h , where h is the water depth

65 Subroutine FSFLOW2

This subroutine is used to compute the fluxes through various types of boundaries and the increasing rate of water in the region of interest. FRATE(7) is to store the flux through the whole boundary enclosing the region of interest. It is given by

$$FRATE(7) = \int_B (V_x n_x + V_y n_y) dB \quad (2.30)$$

where B is the global boundary of the region of interest; V_x and V_y are the flow velocity components; and n_x and n_y are the directional cosines of the outward unit vector normal to the boundary B . FRATE(1) and FRATE(2) store the flux through Dirichlet boundary B_D and Cauchy boundary B_C , respectively, and are given by:

$$FRATE(1) = \int_{B_D} h (V_x n_x + V_y n_y) dB \quad (2.31)$$

$$\text{FRATE}(2) = \int_{B_C} h(V_x n_x + V_y n_y) dB \quad (2.32)$$

FRATE(6), which is related to the numerical loss, is given by

$$\text{FRATE}(6) = \text{FRATE}(7) - \sum_{I=1}^2 \text{FRATE}(I) \quad (2.33)$$

FRATE(8) and FRATE(9) are used to store the rainfall and increased rate of water within the media, respectively:

$$\text{FRATE}(8) = - \int_R (r-i) dR \quad (2.34)$$

and

$$\text{FRATE}(9) = \int_R \frac{\partial h}{\partial t} dR \quad (2.35)$$

If there is no numerical error in the computation, the following equation should be satisfied:

$$\text{FRATE}(9) = - [\text{FRATE}(7) + \text{FRATE}(8)] \quad (2.36)$$

and FRATE(6) should be equal to zero. Equation (2.36) simply states that the negative rate of water going out from the region through the entire boundary and due to source/sink is equal to the rate of water accumulated in the region. This subroutine is called by the program MAIN.

66 Subroutine Q4TH and Q3TH

These subroutines are used to compute the contribution of water increasing rate from an element e:

$$\text{QTHP} = \int_{R_e} \frac{\partial h}{\partial t} dR \quad (2.37)$$

The computation of the above integration is straightforward. Q4TH is used to evaluate the integration for quadrilateral elements and Q3TH for triangular elements.

67 Subroutine FFLUX

This subroutine is called by subroutine FSFLOW2 to calculate the flux through each boundary point on the 2-D overland region.

68 Subroutine FPRINT2

This subroutine is used to line-print the flow variables. These include the mass balance table, the water depth, and the velocity components.

69 Subroutine FSTORE2

This subroutine is called by the program MAIN to save water depth, velocity, or infiltration rate at each nodal point on 2-D overland for use of GMS post-processor.

70 Subroutine FSTORE2B

This subroutine is used to store the flow variables on Logical Unit 11. It is intended for use with a subsequent computer model or for plotting. The information stored includes region geometry, and hydrological variables such as water depth and velocity components.

71 Subroutine DVRGZ0

This subroutine computes the following equation at each point

$$\sqrt{1 + \left(\frac{\partial Z_0}{\partial x}\right)^2} \times \sqrt{1 + \left(\frac{\partial Z_0}{\partial y}\right)^2} \quad (2.38)$$

used for the computation of infiltration rate in subroutine INFNOD2. Z_0 is the elevation of that point.

72 Subroutine D34DZ0

This subroutine is called by subroutine DVRGZ0 to compute the following equations.

$$QB(I,J) = \int_{R_e} N_i^e N_j^e dR \quad (2.39)$$

where N_i^e and N_j^e are the basis functions for nodal point i and j of element e, respectively. Subroutines Q4D and Q3D also evaluate the element load vector:

$$QRX(I) = - \int_{R_e} N_i^e \frac{\partial N_j^e}{\partial x} Z_{0j} dR \quad (2.40)$$

$$QRY(I) = - \int_{R_e} N_i^e \frac{\partial N_j^e}{\partial y} Z_{0j} dR \quad (2.41)$$

where

Z_{0j} = the ground-surface elevation at nodal point j;

73 Subroutine GAUPNT2

This subroutine is called by subroutine D34DZ0 to calculate the base functions at Gaussian points.

74 Subroutine INFNOD2

This subroutine is called by program MAIN to compute infiltration flux (L^3/T) at each nodal point on the 2-D overland domain. That is, the following equation is obtained based on the concept of continuity of flux on the ground surface and coded in this subroutine.

$$\int_R -(n_x V_x + n_y V_y + n_z V_z) \times \sqrt{1 + \left(\frac{\partial Z_0}{\partial x}\right)^2} \times \sqrt{1 + \left(\frac{\partial Z_0}{\partial y}\right)^2} dR \quad (2.42)$$

This information is then used by 2-D overland and 3-D subsurface flow simulations.

75 Subroutine INF3R

This subroutine is called by program MAIN to compute the groundwater velocities at the very beginning of the 2-D overland flow simulation. The normal velocities on the interface between the retention pond and 3-D subsurface are the flux rate for the computation of $QS(j)$ in Eq. (2.14). This subroutine calls subroutines ELENOD3 and FQ468DV to compute the elemental lumped matrix.

76 Subroutine HYDROR2

This subroutine is called by program MAIN to control the time loop simulation of retention ponds and 2-D overland flow in a subsurface flow time step. It calls subroutines INTRP to obtain the rainfall rate, or to calculate the boundary values, RPOND to compute the water depth of retention ponds using water budget concept, and OF2D to determine the water depth on the ground surface at an overland flow time step. The flux rate through each intra-boundary between retention pond and dike as well as 2-D overland and 3-D subsurface is also accumulated and saved for use of 3-D subsurface flow simulation.

77 Subroutine RPOND

This subroutine is called by subroutine HYDROR2 to compute the water depth in retention ponds. According to the water budget concept, Eq. (2.14) is therefore coded in this subroutine. One must be aware that the water elevation is considered horizontal in the retention pond so that the water budget concept can be implemented.

78 Subroutine NHRPON

This subroutine is called by RPOND to calculate the new water surface elevation of water in retention ponds. The required information for this subroutine is the total water coming into the retention pond and the water surface elevation at the previous time. It also determined the total flux flowing out through the outlet on the retention pond.

79 Subroutine OF2D

The subroutine OF2D will perform a transient state computation. It calls subroutine SPROP2 to obtain the K value; subroutine VELT2 to compute the flow velocity; subroutine FASEMB2 to assemble the element matrices over all elements; subroutine FBC2 to implement the boundary conditions; and subroutine BANSOL, PISS, and PPCG to solve the linearize matrix equation.

80 Subroutine FASEMB2

This subroutine calls Q4 and Q3 to evaluate the element matrices. It then sums over all element matrices to form a global matrix equation governing the water depth at all nodes. The volumetric infiltration rate is considered as a point sink and subtracted from the load vector of the matrix equation.

81 Subroutine Q4 and Q3

This subroutine is called by the subroutine FASEMB2 to compute the element matrix given by:

$$QA(I,J) = \int_{R_e} N_i^e N_j^e dR \quad (2.43)$$

$$QB(I,J) = \int_{R_e} (\nabla N_i^e) \cdot K \cdot (\nabla N_j^e) dR \quad (2.44)$$

where F is the soil property function. Q4 is to compute Eqs. (2.43) and (2.44) for quadrilateral elements, while Q3 is to compute those for triangular elements. Subroutines Q4 and Q3 also calculate the element load vector given by:

$$RQ(I) = \int_{R_e} [(\nabla N_i^e) \cdot K \cdot (\nabla z) - N_i^e r] dR \quad (2.45)$$

where r is the rainfall rate.

82 Subroutine BASE0

This subroutine is called by subroutines SPROP2, Q4D, Q4TH, Q4, and Q34S to evaluate the value of the base function at a Gaussian point. The computation is straightforward.

83 Subroutine FBC2

This subroutine incorporates Dirichlet, Cauchy, or Radiation boundary conditions. For a Dirichlet boundary condition, an identity algebraic equation is generated for each Dirichlet nodal point. Any other equation having this nodal variable is modified accordingly to simplify the computation. For a Cauchy or radiation surface, the integration of the surface source is added to the load vector.

84 Subroutine Q2FB

This subroutine is called by the subroutine FBC2 to compute the surface integrals of the following types

$$BQ(I,J) = - \int_{B_e} N_i f_b N_j^e dB, \quad (2.46)$$

$$RQ(I) = - \int_{B_e} N_i f_q dB, \quad (2.47)$$

where f_q and f_b are function of boundary conditions. They are determined in the subroutine FBC2. For a Cauchy or radiation boundary condition, f_q represents the Cauchy or radiation flux and $f_b = 0$.

85 Subroutine BANSOL

This subroutine is called by the subroutines VELT2 and OF2D to solve for the matrix equation of the type:

$$[C] \{x\} = \{y\} \quad (2.48)$$

where $[C]$ is the coefficient matrix and $\{x\}$ and $\{y\}$ are two vectors. $\{x\}$ is the unknown to be solved and $\{y\}$ is the known load vector. The computer returns the solution $\{x\}$ and stores it in $\{y\}$. The computation is a standard banded Gaussian direct elimination procedure.

86 Subroutine PISS

This subroutine is called by the subroutine OF2D and SF3D, if necessary, to solve the linearized matrix equation with pointwise iteration solution strategies.

87 Subroutine PPCG

This subroutine is called by the subroutine OF2D and SF3D, if necessary, to solve the linearized matrix equation with the preconditioned conjugate gradient method using the polynomial as a preconditioner. It calls POLYP to invert the preconditioner.

88 Subroutine POLYP

This subroutine is called by the subroutine PPCG to solve for a modified residual that will be used in the preconditioned conjugate gradient algorithm.

89 Subroutine ILUCG

This subroutine is called by the subroutine OF2D and SF3D, if necessary, to solve the linearized matrix equation with the preconditioned conjugate gradient method using the incomplete Cholesky decomposition as a preconditioner. It calls to LLTINV to invert the preconditioner.

90 Subroutine LLTINV

This subroutine is called by the subroutine ILUCG to solve for a modified residual that will be used in the preconditioned conjugate gradient algorithm.

91 Subroutine SPROP3

This subroutine calculates the values of moisture content, relative hydraulic conductivity, and the water capacity. Either tabular input or analytical functions can be used to represent soil property functions. When analytical functions are used, the users must supply the functional form.

92 Subroutine SHAPE

This subroutine is called by subroutines SPROP3, FQ468DV, FQ468, and Q468TH to evaluate the value of the base and weighting functions and their derivatives at a Gaussian point. The computation is straightforward.

93 Subroutine SF3D

SF3D calls subroutine SPROP3 to obtain the relative hydraulic conductivity, water capacity, and moisture content from the pressure head; subroutine VELT3 to compute Darcy's velocity; subroutine BCPREP to determine if a change of boundary conditions is required; subroutines BCTOP and BCF3R to check if a switch between Dirichlet and Flux type of boundary conditions implemented on the ground surface and on the bottom surface of the retention pond, respectively is required; subroutine FASEMB3 to assemble the element matrices over all elements; subroutine FBC3 to implement the boundary conditions; subroutine BLKITR, PISS, PPCG, or ILUCG to solve the matrix equations.

94 Subroutine BCPREP

This subroutine is called by SF3D to prepare the infiltration-seepage boundary conditions during a rainfall period or the seepage-evapotranspiration boundary conditions during non-rainfall periods. It decides the number of nodal points on the Variable boundary to be considered as Dirichlet or Cauchy points. It computes the number of points that change boundary conditions from ponding depth (Dirichlet types) to infiltration (Cauchy types), or from infiltration to ponding depth, or from minimum pressure (Dirichlet types) to infiltration during rainfall periods. It also computes the number of points that change boundary conditions

from potential evapotranspiration (Cauchy types) to minimum pressure, or from ponding depth to potential evapotranspiration, or from minimum pressure to potential evapotranspiration during non-rainfall periods. Upon completion, this subroutine returns the Darcy flux (DCYFLX), infiltration/potential evapotranspiration rate (FLX), the ponding depth nodal index (NPCON), the flux-type nodal index (NPFLX), the minimum pressure nodal index (NPMIN), and the number of nodal points (NCHG) that have changed boundary conditions.

95 Subroutine BCTOP

This subroutine is called by SF3D to determine the type of boundary condition (either the flux type or Dirichlet type) when coupled surface and subsurface flow is simulated. It decides the nodal number on the top boundary to be considered as Dirichlet or flux points. It computes the number of points that change boundary conditions from Dirichlet types to infiltration (flux types), or from infiltration to Dirichlet type. It returns the number of nodal points (NBCHG) that have changed boundary conditions.

96 Subroutine BCF3R

This subroutine is called by SF3D to determine the type of boundary condition (either the flux type or Dirichlet type) to apply to the bottom surface of the retention pond when coupled surface and subsurface flow is simulated. It decides the nodal number on the top boundary to be considered as Dirichlet or flux points. It computes the number of points that change boundary conditions from Dirichlet types to infiltration (flux types), or from infiltration to Dirichlet type. It returns the number of nodal points (N3RCHG) that have changed boundary conditions.

97 Subroutine Q34S

This subroutine is called by the subroutines BCPREP, FBC3, and FSFLOW3 to compute the surface node flux of the type

$$RQ(I) = \int_{B_e} N_i^e q dB , \quad (2.49)$$

where q is either the Cauchy flux, Neumann flux, or gravity flux, or the seepage rate.

98 Subroutine FASEMB3

This subroutine calls FQ468 to evaluate the element matrices. It then sums over all element matrices to form a global matrix equation governing the pressure head at all nodes. The volumetric infiltration rate is considered a point source and added to the load vector of the matrix equation.

99 Subroutine FQ468

This subroutine is called by the subroutine FASEMB3 to compute the element matrix given by

$$QA(I,J) = \int_{R_e} N_i^e F N_j^e dR , \quad (2.50)$$

$$QB(I,J) = \int_{R_e} (\nabla N_i^e) \cdot \mathbf{K} \cdot (\nabla N_j^e) dR , \quad (2.51)$$

where F is defined in Eq. (2.1). Subroutine FQ468 also calculates the element load vector given by

$$RQ(I) = \int_{R_e} [(\nabla N_i^e) \cdot \mathbf{K} \cdot \nabla z - N_i^e q] dR , \quad (2.52)$$

where q is the source/sink.

100 Subroutine FBC3

This subroutine incorporates Dirichlet, Cauchy, Neumann, and Variable boundary conditions. For a Dirichlet boundary condition, an identity algebraic equation is generated for each Dirichlet nodal point. Any other equation having this nodal variable is modified accordingly to simplify the computation. For a Cauchy surface, the integration of the surface source is obtained by calling the subroutine Q34S, and the result is added to the load vector. For a Neumann surface, the integrations of both the gradient and gravity fluxes are obtained by calling the subroutine Q34S. These fluxes are added to the load vector. The subroutine FBC3 also

implements the Variable boundary conditions. First, it checks all infiltration-evapotranspiration-seepage points, identifying any of them that are Dirichlet points. If there are Dirichlet points, the method of incorporating Dirichlet boundary conditions mentioned above is used. If a given point is not the Dirichlet point, the point is bypassed. Second, it checks all rainfall-evaporation-seepage points again to see if any of them is a Cauchy point. If it is a Cauchy point, then the computed flux by infiltration or potential evapotranspiration is added to the load vector. If a given point is not a Cauchy point, it is bypassed. Because the infiltration-evaporation-seepage points are either Dirichlet or Cauchy points, all points are taken care of in this manner.

101 Subroutine Q34R

This subroutine is called by the subroutines FBC3 to implement River boundary conditions, hence to compute the surface node flux of the type

$$RQ(I) = \int_{B_e} N_i^e \frac{K_R}{b_R} h_R dB , \quad (2.53)$$

$$BQ(I,j) = \int_{B_e} N_i^e \frac{K_R}{b_R} N_j^e dB , \quad (2.54)$$

102 Subroutine BLKTR

This subroutine is called by the subroutine SF3D to solve the matrix equation with block iteration methods. For each subregion, a block matrix equation is constructed based on the global matrix equation and two pointer arrays GNPLR and LNOJCN (see subroutine PAGEN3), and the resulting block matrix equation is solved with the direct band matrix solver by calling subroutine SOLVE. This is done for all subregions for each iteration until a convergent solution is obtained. This subroutine and the subroutine SOLVE, to be described in the next paragraph, are needed only for when the block iteration option is used.

103 Subroutine SOLVE

This subroutine is called by the subroutine BLKITER to solve for the matrix equation of the type

$$[C]\{x\} = \{y\} \quad (2.55)$$

where $[C]$ is the coefficient matrix and $\{x\}$ and $\{y\}$ are two vectors. $\{x\}$ is the unknown to be solved, and $\{y\}$ is the known load vector. The computer returns the solution $\{x\}$ and stores it in $\{y\}$. The computation is a standard banded Gaussian direct elimination procedure.

104 Subroutine VELT3

This subroutine calls FQ468DV to evaluate the element matrices and the derivatives of the total head. It then sums over all element matrices to form a matrix equation governing the velocity components at all nodal points. To save computational time, the matrix is diagonalized by lumping. The velocity components can thus be solved point by point. The computed velocity field is then returned to program MAIN or SF3D through the argument. This velocity field is also passed to subroutine BCPREP to evaluate the Darcy flux across the seepage-infiltration-evapotranspiration surfaces and to subroutine INFNOD2 to evaluate infiltration rate.

105 Subroutine FQ468DV

Subroutine FQ468DV is called by the subroutine VELT3 to compute the element matrices given by

$$QR(I,J) = \int_{B_e} N_i^e N_j^e dR, \quad (2.56)$$

where N_i^e and N_j^e are the basis functions for nodal point i and j of element e , respectively. Subroutine FQ468DV also evaluates the element load vector:

$$QRX(I) = - \int_{R_e} N_i^e \mathbf{i} \cdot \mathbf{K} \cdot (\nabla N_j^e) h_j dR - \int_{R_e} N_i^e \mathbf{i} \cdot \mathbf{K} \cdot \nabla z dr \quad (2.57)$$

$$QRY(I) = - \int_{R_e} N_i^e \mathbf{j} \cdot \mathbf{K} \cdot (\nabla N_j^e) h_j dR - \int_{R_e} N_i^e \mathbf{j} \cdot \mathbf{K} \cdot \nabla z dr \quad (2.58)$$

$$QRZ(I) = - \int_{R_e} N_i^e \mathbf{k} \cdot \mathbf{K} \cdot (\nabla N_j^e) h_j dR - \int_{R_e} N_i^e \mathbf{k} \cdot \mathbf{K} \cdot \nabla z dr \quad (2.59)$$

where

h_j = the referenced pressure head at nodal point j ,

\mathbf{i} = the unit vector along the x -coordinate,

\mathbf{j} = the unit vector along the y -coordinate,

\mathbf{k} = the unit vector along the z -coordinate,

\mathbf{K}_s = the saturated hydraulic conductivity tensor,

K_r = the relative hydraulic conductivity.

106 Subroutine FSFLOW3

This subroutine is used to compute the fluxes through various types of boundaries and the increasing rate of water content in the region of interest. The function of FRATE(7) is to store the flux through the whole boundary enclosing the region of interest. It is given by

$$\text{FRATE}(7) = \int_B (V_x n_x + V_y n_y + V_z n_z) dB, \quad (2.60)$$

where B is the global boundary of the region of interest; V_x , V_y , and V_z are Darcy's velocity components; and n_x , n_y , and n_z are the directional cosines of the outward unit vector normal to the boundary B . FRATE(1) through FRATE(5) store the flux through Dirichlet boundary B_D , Cauchy boundary B_C , Neumann boundary B_N , the seepage/evapotranspiration boundary B_S , and infiltration boundary B_R , respectively, and are given by

$$\text{FRATE}(1) = \int_{B_D} (V_x n_x + V_y n_y + V_z n_z) dB, \quad (2.61)$$

$$\text{FRATE}(2) = \int_{B_C} (V_x n_x + V_y n_y + V_z n_z) dB, \quad (2.62)$$

$$\text{FRATE}(3) = \int_{B_N} (V_x n_x + V_y n_y + V_z n_z) dB, \quad (2.63)$$

$$\text{FRATE}(4) = \int_{B_S} (V_x n_x + V_y n_y + V_z n_z) dB , \quad (2.64)$$

$$\text{FRATE}(5) = \int_{B_R} (V_x n_x + V_y n_y + V_z n_z) dB , \quad (2.65)$$

FRATE(6), which is related to the numerical loss, is given by

$$\text{FRATE}(6) = \text{FRATE}(7) - \sum_{I=1}^5 \text{FRATE}(I) \quad (2.66)$$

FRATE(8) and FRATE(9) are used to store the source/sink and increased rate of water within the media, respectively:

$$\text{FRATE}(8) = - \int_R q dR , \quad (2.67)$$

and

$$\text{FRATE}(9) = \int_R F \frac{\partial h}{\partial t} dR , \quad (2.68)$$

If there is no numerical error in the computation, the following equation should be satisfied:

$$\text{FRATE}(9) = -[\text{FRATE}(7) + \text{FRATE}(8)] \quad (2.69)$$

and FRATE(6) should be equal to zero. Equation (2.69) simply states that the negative rate of water going out from the region through the entire boundary and due to a source/sink is equal to the rate of water accumulated in the region.

107 Subroutine Q468TH

This subroutine is used to compute the contribution of the increasing rate of the water content from an element e

$$QTHP = \int_{R_e} F \frac{\partial h}{\partial t} dR , \quad (2.70)$$

The computation of the above integration is straightforward.

108 Subroutine FPRINT3

This subroutine is used to line-print the flow variables. These include the fluxes through Variable boundary surfaces, the pressure head, total head, moisture content, and Darcy's velocity components.

109 Subroutine THNPON

This subroutine is used to calculate the moisture content at nodal points based on that computed at quadrature points. The result is saved, if requested, for plotting purpose.

110 Subroutine FSTORE3

This subroutine is used to save the results regarding pressure head, total head, moisture content at nodal points, and Darcy velocity for use of GMS post-processor. The associated filenames must be provided in the super file for the simulation.

111 Subroutine FSTORE3B

This subroutine is used to store the flow variables on Logical Unit 11. It is intended for use for plotting. The information stored includes region geometry, subregion data, and hydrological variables such as pressure head, total head, moisture content, and Darcy's velocity components.

112 Subroutine CSTORE

This subroutine is called by the program MAIN to store the result of canal simulation for use of GMS post-processor. The associated filename must be provided in the super file for the simulation.

113 Subroutine DISQ

This subroutine is called by the program MAIN to calculate the flux through each specified cross section at each time step. The result is then saved for printing and plotting the hydrograph.

114 Subroutine PRINTQ

This subroutine is called by the program MAIN to print the hydrograph data at each time step.

115 Subroutine QSTORE

This subroutine is called by the program MAIN to store the result of hydrograph data for use of GMS post-processor.

116 Subroutine PRINRP

This subroutine is called by the program MAIN to write out the water surface elevation of each retention pond at each time step.

117 Subroutine RPSTOR

This subroutine is called by the program MAIN to store the computation result for retention pond simulation for use of GMS post-processor at each time step.

3. ADAPTATION OF COSFLOW TO SITE SPECIFIC APPLICATIONS

The following describes how one should apply the COSFLOW code for site-specific applications and how the data file should be prepared.

3.1 Parameters Specifications

For each site-specific problem, the users only need to specify the size of the problem by assigning 151 maximum control-integers with PARAMETER STATEMENT in the including file (with the extension name .inc). The list and definitions of the maximum control-integers required for both flow and transport simulations are given below even though only water flow in the coupled system is centered in this version.

Maximum Control-Integers for the Subsurface Spatial Domain

MXSNP = maximum no. of subsurface nodes;
 MXSEL = maximum no. of subsurface elements,
 MXSBNP = maximum no. of boundary nodal points in the 3-D subsurface;
 MXSBES = maximum no. of boundary-element surfaces in the 3-D subsurface,
 MXSJBD = maximum no. of nonzero elements in any row for nodewise connectivity;
 MXSKBD = maximum no. of nonzero elements in any row for elementwise connectivity.

Maximum Control-Integers for the Groundwater Time Domain

MXSNTI = maximum no. of time steps for the simulation of 3-D subsurface flow,
 MXSDTC = maximum no. of DELT changes for the simulation of 3-D subsurface flow;

Maximum Control-Integers for Subregions in the 3-D Subsurface

LTMXSNP = maximum no. of total nodal points in any subregion, including interior nodes, global boundary nodes, and intraboundary nodes. LTMXSNP = 1 if the block iteration is not used.
 LMXSNP = maximum no. of nodal points in any subregion, including interior nodes and global boundary nodes. LMXSNP = 1 if the block iteration is not used.
 LMXSBW = maximum no. of the bandwidth in any subregion. LMXSBW = 1 if the block iteration is not used.
 MXSREGN = maximum no. of subregions. MXSREGN = 1 if the block iteration is not used.

Maximum Control-Integers for Material and Soil Properties in the 3-D Subsurface

MXSMAT = maximum no. of material types,
 MXSSPPM = maximum no. of soil parameters per material to describe soil characteristic curves,
 MXSMPPM = maximum no. of material properties per material;
 MXSRMP = maximum no. of coefficients of the viscosity and density functions.

The maximum control-integers for 3-D subsurface flow simulations and their definitions are given as the following:

Maximum Control-Integers for Source/sinks, 3-D subsurface flow

MXSSELF = maximum no. of source elements,
MXSSPRF = maximum no. of source profiles,
MXSSDPF = maximum no. of data points on each element source/sink profile,
MXSWNPF = maximum no. of well nodal points,
MXSWPRF = maximum no. of well source/sink profiles,
MXSWDPF = maximum no. of data points on each well source/sink profile;

Maximum Control-Integers for Cauchy Boundary Conditions, 3-D subsurface flow

MXSCNPF = maximum no. of Cauchy nodal points,
MXSCESF = maximum no. of Cauchy element surfaces,
MXSCPRF = maximum no. of Cauchy-flux profiles,
MXSCDPF = maximum no. of data points on each Cauchy-flux profile;

Maximum Control-Integers for Neumann Boundary Conditions, 3-D subsurface flow

MXSNNPF = maximum no. of Neumann nodal points,
MXSNESF = maximum no. of Neumann element surfaces,
MXSNPRF = maximum no. of Neumann-flux profiles,
MXSNDPF = maximum no. of data points on each Neumann-flux profile;

Maximum Control-Integers for Rainfall-Seepage Boundary Conditions, 3-D subsurface flow

MXSVNPF = maximum no. of Variable nodal points,
MXSVESF = maximum no. of Variable element surfaces,
MXSVPRF = maximum no. of rainfall profiles,
MXSVDPF = maximum no. of data point on each rainfall profile;

Maximum Control-Integers for Dirichlet Boundary Conditions, 3-D subsurface flow

MXSDNPF = maximum no. of Dirichlet nodal points,
MXSDPRF = maximum no. of Dirichlet total head profiles,
MXSDDPF = maximum no. of data points on each Dirichlet profile;

Maximum Control-Integers for River Boundary Conditions, 3-D subsurface flow

MXSRNPF = maximum no. of River nodal points,
MXSRPRF = maximum no. of River profiles,
MXSRDPF = maximum no. of data point on each River profile,
MXSRMAF = maximum no. of River boundary materials;
MXSRESF = maximum no. of River element surfaces.

The maximum control-integers for transport simulations and their definitions are given as the

following:

Maximum Control-Integers for Source/sinks, transport

MXSSELT = maximum no. of source elements,
 MXSSPRT = maximum no. of source profiles,
 MXSSDPT = maximum no. of data points on each element source/sink profile,
 MXSWNPT = maximum no. of well nodal points,
 MXSWPRT = maximum no. of well source/sink profiles,
 MXSWDPT = maximum no. of data points on each well source/sink profile;

Maximum Control-Integers for Cauchy Boundary Conditions, transport

MXSCNPT = maximum no. of Cauchy nodal points,
 MXSCEST = maximum no. of Cauchy element surfaces,
 MXSCPRT = maximum no. of Cauchy-flux profiles,
 MXSCDPT = maximum no. of data points on each Cauchy-flux profile;

Maximum Control-Integers for Neumann Boundary Conditions, transport

MXSNNPT = maximum no. of Neumann nodal points,
 MXSNEST = maximum no. of Neumann element surfaces,
 MXSNPRT = maximum no. of Neumann-flux profiles,
 MXSNDPT = maximum no. of data points on each Neumann-flux profile;

Maximum Control-Integers for Flowin-Flowout Boundary Conditions, transport

MXSVNPT = maximum no. of variable nodal points,
 MXSVEST = maximum no. of variable element surfaces,
 MXSVPRT = maximum no. of rainfall profiles,
 MXSVDPT = maximum no. of data point on each rainfall profile;

Maximum Control-Integers for Dirichlet Boundary Conditions, transport

MXSDNPT = maximum no. of Dirichlet nodal points,
 MXSDPRT = maximum no. of Dirichlet concentration profiles,
 MXSDDPT = maximum no. of data points on each Dirichlet profile;

Maximum Control-Integers for Particle Tracking, transport

MXSNPW = maximum no. of nodal points used for in-element particle tracking,
 MXSELW = maximum. no. of working elements used for in-element particle tracking.

The above 24 control-integers are used for the subsurface transport module, they all therefore can be set to 1 in the simulation of coupling 1-D channel flow, 2-D overland flow, and 3-D subsurface flows.

The maximum control-integers for 1-D canal flow simulations and their definitions are given as the following:

Maximum Control-Integers for 1-D Canal Flow

MXCNRN = maximum no. of channel reach nodes,
 MXCNR = maximum no. of channel reaches,
 MXCEL = maximum no. of channel elements,
 MXCNP = maximum no. of channel element nodes,
 MXCCPM = maximum no. of channel pumps,
 MXCGPR = maximum no. of flow rate profiles for interior gates,
 MXCGPD = maximum no. of data points on each rating curve that is associated with interior gate profiles,
 MXCWPR = maximum no. of flow rate profiles for weir,
 MXCWPD = maximum no. of data points on each rating curve that is associated with weir profiles,
 MXCRPR = maximum no. of profiles for channel precipitation rate,
 MXCRPD = maximum no. of data points on each channel precipitation rate profile,
 MXCPPR = maximum no. of profiles for channel pumping rate,
 MXCPPD = maximum no. of data points on each channel pumping rate profile,
 MXCUPR = maximum no. of flow rate profiles for upstream boundary gates,
 MXCUPD = maximum no. of data points on each rating curves rating curve that is associated with upstream boundary gate profiles,
 MXCDPR = maximum no. of profiles for downstream boundary gates,
 MXCDPD = maximum no. of data points on each rating curve that is associated with downstream boundary gate profiles,
 MXCNTC = maximum no. of channel time steps allowed within one subsurface time step,
 MXCNJR = maximum no. of groups of channels that are connected by structure-free joints,
 MXCANG = maximum no. of data points to describe the head of Angel Well in South Florida,
 MXCHMXR = maximum no. of profiles for the upper stage limit of channel reaches for a control purpose,
 MXCHMXD = maximum no. of data points on each profile for the channel upper stage limit,
 MXCHMNR = maximum no. of profiles for the lower stage limit of channel reaches for a control purpose,
 MXCHMND = maximum no. of data points on each profile for the channel lower stage limit,
 MXCSNP = maximum no. of subsurface nodes that are on the subsurface-channel interface,
 MXCSEL = maximum no. of subsurface boundary sides to compose the subsurface-channel interface.
 MXCNGO = maximum no. of opening status associated with the gate profiles.

The maximum control-integers for 2-D overland flow simulations and their definitions are given as the following:

Maximum Control-Integers for the 2-D Overland Spatial Domain

MXONP = maximum no. of nodes in the 2-D overland region,
 MXOEL = maximum no. of elements in the 2-D overland region,
 MXOBNP = maximum no. of boundary nodal points in the 2-D overland region
 MXOBES = maximum no. of boundary-element surfaces in the 2-D overland region,
 MXOHBP = maximum no. of half band width plus 1,
 MXOJBD = maximum no. of nodes connecting to any node,

MXOKBD = maximum no. of elements connecting to any node;

Maximum Control-Integers for the 2-D Overland Time Domain

MXOVTK = maximum no. of channel time steps included in a overland flow time step,

MXONTI = maximum no. of time steps for the simulation of 2-D overland flow,

MXODTC = maximum no. of DELT changes for the simulation of 2-D overland flow;

Maximum Control-Integers for Source/sinks of 2-D Overland Flow Simulation

MXORES = MXOEL, maximum no. of elements prescribed by a rainfall profile,

MXORPR = maximum no. of source profiles,

MXORDP = maximum no. of data points on each element source/sink profile;

Maximum Control-Integers for Dirichlet Boundary Conditions of 2-D Overland Flow Simulation

MXODNP = maximum no. of Dirichlet nodal points,

MXODPR = maximum no. of Dirichlet water depth profiles,

MXODDP = maximum no. of data points on each Dirichlet profile;

Maximum Control-Integers for Cauchy Boundary Conditions of 2-D Overland Flow Simulation

MXOCNP = maximum no. of Cauchy nodal points,

MXOCES = maximum no. of Cauchy element surfaces,

MXOCPR = maximum no. of Cauchy-flux profiles,

MXOCDP = maximum no. of data points on each Cauchy-flux profile;

Maximum Control-Integers for Radiation Boundary Conditions of 2-D Overland Flow Simulation

MXONNP = maximum no. of radiation boundary nodes,

MXONES = maximum no. of radiation boundary element surfaces,

MXONPR = maximum no. of radiation rating curves,

MXONDP = maximum no. of data points on each radiation rating curve;

Maximum Control-Integers for Material Properties in the 2-D Overland Flow Region

MXOMAT = maximum no. of material types.

The maximum control-integers for the simulation of retention ponds and their definitions are given as the following:

Maximum Control-Integers for Retention Ponds

MXPONK = maximum no. of retention ponds,

MXPPRK = maximum no. of rainfall profiles,

MXDPPK = maximum no. of data points describing rainfall profiles,

MXPOBK = maximum no. of boundary element sides in between retention pond and dike (subsurface),

MXOULK = maximum no. of boundary element sides on which exist an outlet of the associated retention pond,

MXFXPK = maximum no. of rating curves describing the h vs. q relationship related to spillways of

retention ponds,
MXFXPK = maximum no. of data points describing each rating curve for the description of water spilled through spillways,

The maximum control-integers for the graphic interface between GMS and the computation kernel are defined as follows.

Maximum Control-Integers for Graphic Interface

MXXYPK = maximum no. of data points in each xy series,
MXXYSK = maximum no. of xy series,
MXPNTK = maximum no. of nodes on each section specified for the calculation of hydrograph,
MXLINK = maximum no. of sections required for the calculation of hydrograph,
MXITPK = maximum no. of points in each hydrograph,
MXPSTK = maximum no. of specified times to save the results for plotting purpose.

For the simulation of coupling 3-D subsurface flow, 2-D overland and retention pond flow, and 1-D channel flow, we demonstrate how to specify the above maximum control-integers with PARAMETER STATEMENT in the including file (with an extension name .inc) by the following example sketched in Figure 3.1.

Let us assume that a subsurface domain is discretized by $7 \times 12 \times 6$ nodes and $6 \times 11 \times 5$ elements. In other words, we are discretizing the region with 7 nodes along the longitudinal or x-direction, 12 nodes along the lateral or y-direction, and 6 nodes along the vertical or z-direction. Since we have a total of $7 \times 12 \times 6 = 504$ nodes, the maximum number of nodes is MXSNP = 504. The total number of elements is $6 \times 11 \times 5 = 330$, i.e., MXSEL = 330. For this simple discretization problem, the maximum connecting number to any of the 504 nodes in the region of interest is 27, i.e., MXSJBD = 27, and the maximum connecting number of elements to any of the 504 nodes is 8, i.e., MXSKBD = 8. There will be $6 \times 11 = 66$ element surfaces each on the bottom and top faces of the region, $6 \times 5 = 30$ element-surfaces each on the front and back faces of the

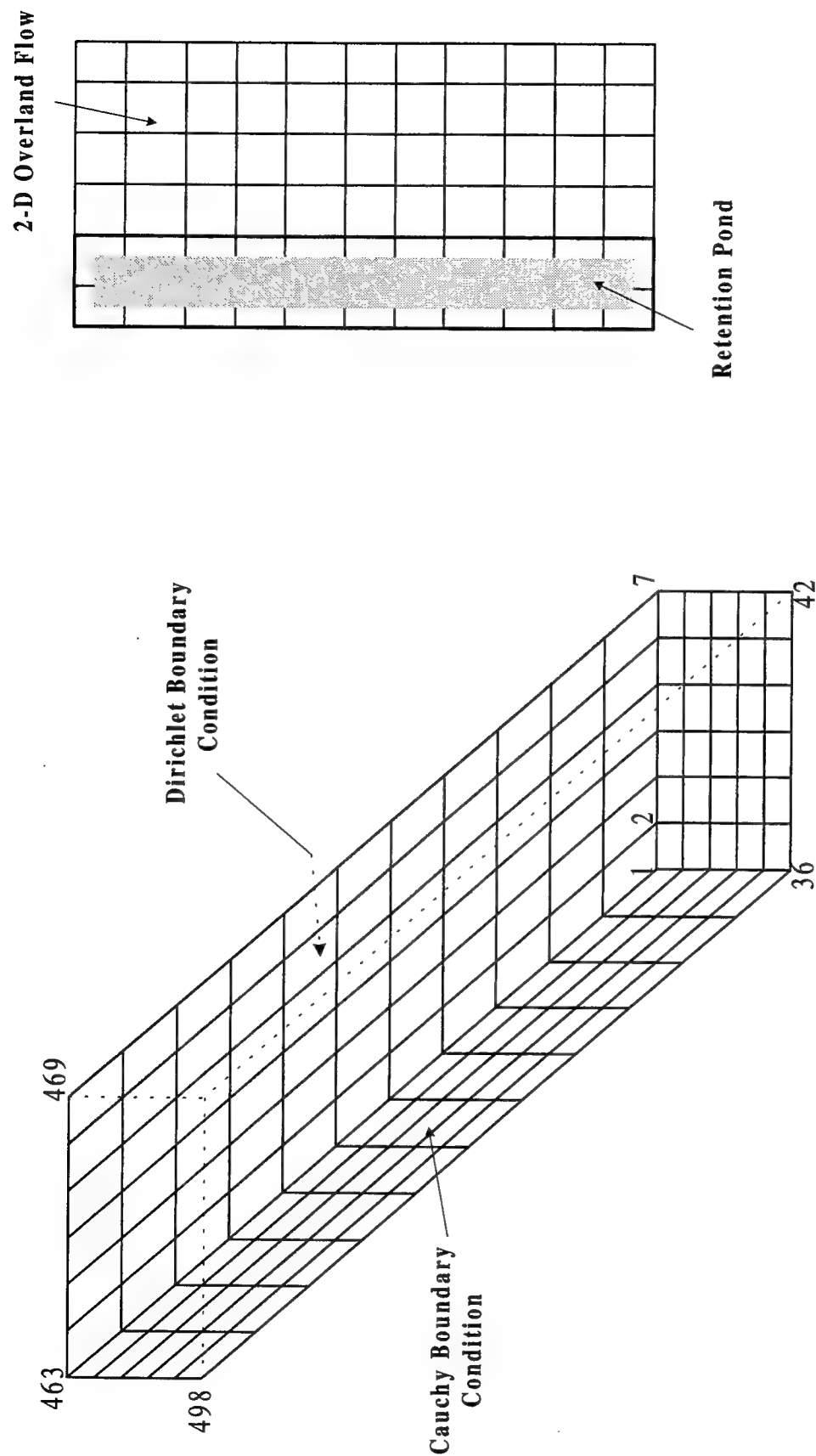


Figure 3.1 The sketch of a simulation problem for demonstration

region, and $11 \times 5 = 55$ element-surfaces each on the left and right faces of the region. Thus, there will be a total of 302 element-surfaces, i.e., MXSBES = 302. Similarly, we can compute the surface-boundary nodes to be 304, i.e., MXSBNP = 304.

In order to specify maximum control-integers related to subregion data, we have to know how the region of interest is subdivided into subregions. Let us assume we have subdivided the region of interest into 12 subregions, each subregion has 7×6 nodes. It is seen, in fact, we are taking a vertical slice as a subregion. For this sub-regionalization, we have MXSREGN = 12. Each subregion has $7 \times 6 = 42$ nodes, resulting LMXSNP = 42. It is also seen that there will be 84 intra-boundary nodes, 42 nodes each on the two neighboring slices of a subregion. Thus, we have LTMXSNP = 126. For each subregion, the maximum bandwidth can be computed as LMXSBW = 17 if the nodes are labeled along the z-directions consecutively.

We will assume that there will be a maximum of 11 elements that have the distributed sources/sinks (i.e., MXSSELF = 11) and a maximum of 10 nodal points that can be considered as well sources/sinks (i.e. MXSWNP = 10, this dose not include the contribution from channel pumping to the subsurface). We will also assume that there will be three different distributed source/sink profiles and five distinct point source/sink profiles. Then we will have MXSSPRF = 3 and MXSWPRF = 5. Let us further assume that four data points are needed to describe the distributed source/sink profiles as a function of time and that 8 data points are required to describe point source/sink profiles (i.e., MXSSDPF = 4 and MXSWDPF = 8).

To specify maximum control-integers for boundary conditions, let us assume that the top face is divided into two parts: 2/3 of it (containing 44 surface elements and 60 nodes) is an interface between 2-D overland and 3-D subsurface and 1/3 of it (including 22 surface elements and 36 nodes) is a retention pond. On the left face, fluxes from the adjacent aquifer are known. On the right face, the total head is assumed known. On the bottom face, natural drainage is assumed to occur (i.e., the gradient of the pressure head can be assumed zero).

There are $12 \times 6 = 72$ nodes on the left face and $11 \times 5 = 55$ element surfaces; thus $MXSCNPF = 72$ and $MXSCESF = 55$. It is further assumed that there two different fluxes going into the region through the left face and that each flux can be described by four data points as a function of time (i.e., $MXSCPRF = 2$, and $MXSCDPF = 4$). On the bottom surface, there are $7 \times 12 = 84$ nodes and $6 \times 11 = 66$ surface elements. Since the gradient of pressure head on the bottom surface is zero, there is only one Neumann flux profile, and two data points, one at zero time and the other at infinite time, are sufficient to describe the constant value of zero. Hence, we have $MXSNNPF = 84$, $MXSNESF = 66$, $MXSNPRF = 1$, and $MXSNDPF = 2$. On the top face, there will be $7 \times 12 = 84$ nodes and $6 \times 11 = 66$ surface elements and 2/3 of it is discretized a 2-D overland flow region (shown in Figure 3.1). Let us assume that there are three different rainfall intensities that might fall on the overland regime, and that each rainfall intensity is a function of time and can be described by 24 data points. With these descriptions, we have $MXONP = 60$, $MXOEL = 44$, $MXORPR = 3$, and $MXORDP = 24$. For the retention pond, we assume one rainfall profile in which 4 data points are included is provided for the rainfall information. Then, we have $MXPONK = 1$, $MXPPRK = 1$, and $MXDPPK = 4$. On the bottom, there are 22 elements and 36 nodes on the interface between retention pond and subsurface. Hence, $MXPOBK = 22$. On the right face, there are $12 \times 6 = 72$ nodes. Let us assume that there are twenty different values of the total head, one each on a vertical line of the right face. We further assume that each of these twenty total head profiles can be described by 8 data points as function of time. We then have $MXSDNPF = 72$, $MXSDPRF = 20$, and $MXSDDPF = 8$.

In this example, we have nine material properties (six saturated hydraulic conductivity components, two compressibility parameters, and one effective porosity) per material. We will assume that the whole region of interest is made of three different kinds of materials. The characteristic curves of each material are assumed to be described by four parameters. We then have $MXSMAT = 3$, $MXSMPPM = 9$, and $MXSSPPM = 4$. If we assume that we will make a 500 time step simulation and we will reinitiate the change on the time

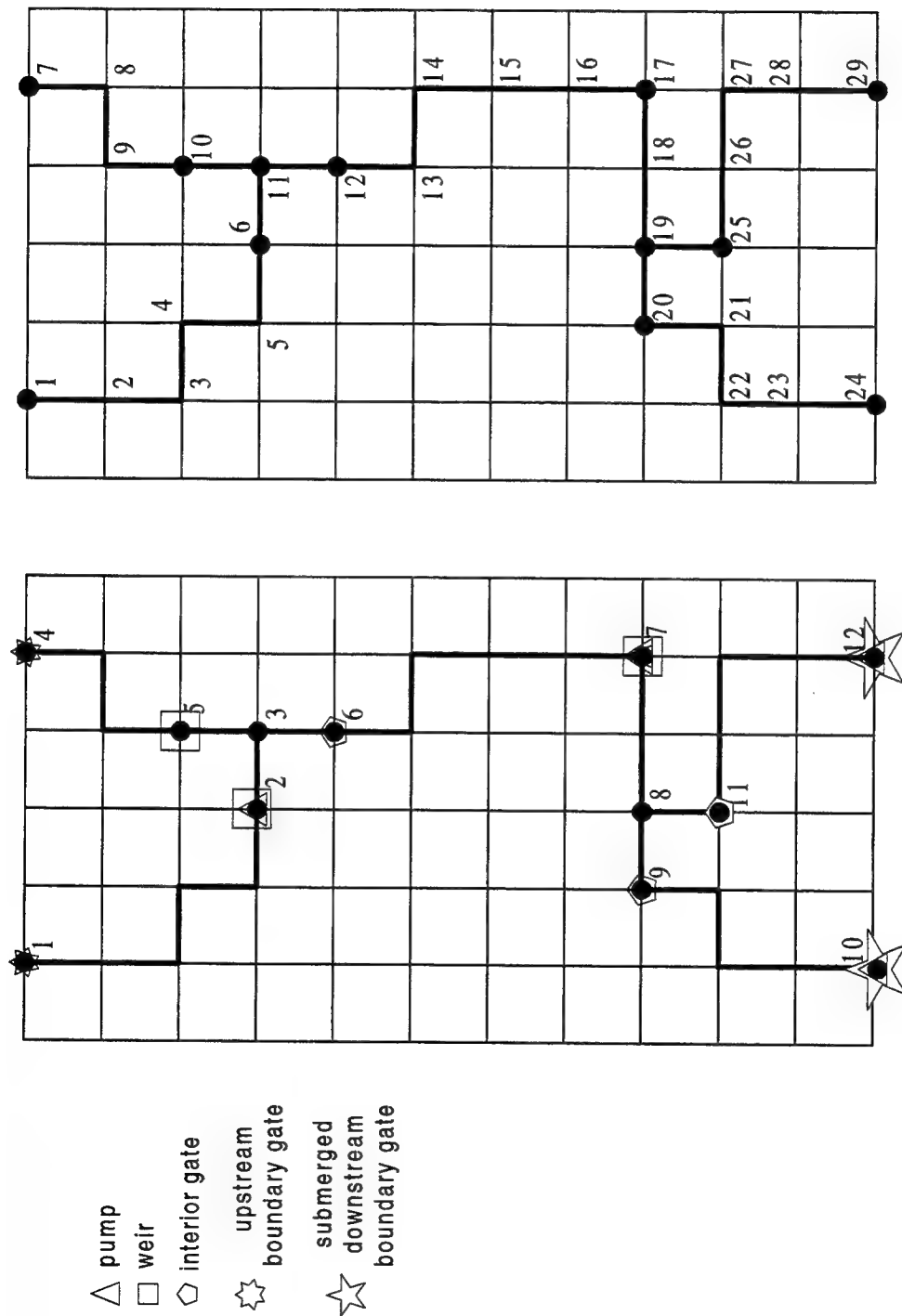
step size for 20 times during our simulation, then we have $\underline{MXSNTI = 500}$ and $\underline{MXSDTC = 20}$.

As mentioned earlier, the control-integers associated with transport module are set to 1 because their associated arrays will not be used in flow simulations.

For the simulation of 1-D channel flow, we suppose there are 11 channel reaches which are connected by 8 channel reach nodes and bounded by another 4 channel reach nodes (shown in Figure 3.2). Therefore, we have $\underline{MXCNRN = 12 (= 8 + 4)}$ and $\underline{MXCNR = 11}$. If these 11 channel reaches are discretized by 28 channel elements and 29 channel element nodes, then we have $\underline{MXCEL = 28}$ and $\underline{MXCNP = 29}$. If 5 channel pumps are installed to adjust water head in channels, we have $\underline{MXCCPM = 5}$. If each pump has its own time-dependent pumping rate profile and each profile is described by 4 data points, then we have $\underline{MXCPPR = 5}$ and $\underline{MXCPPD = 4}$.

Among the 12 channel reach nodes, we assume that there are three interior gates, three weirs, two structure-free joints, two upstream boundary gates, and two downstream boundary gates. Therefore, we have $\underline{MXCGPR = 3}$, $\underline{MXCWPR = 3}$, $\underline{MXCUPR = 2}$, and $\underline{MXCDPR = 2}$. There are two rating curves, Q vs. Δh for the submerged case and Q vs. h for the non-submerged case, to describe a flow rate profile of interior gates or weirs. For upstream boundary gates, each profile contains only one rating curve: Q vs. t . For the case of downstream boundary gates, a profile includes three rating curves: Q vs. ΔH for submerged flow, Q vs. h for non-submerged flow, and t vs. H for the tailwater. If all the rating curves used in gate and weir profiles need 10 data points for the description, then we have $\underline{MXCGPD = 10}$, $\underline{MXCWPD = 10}$, $\underline{MXCUPD = 10}$, and $\underline{MXCDPD = 10}$. We also provide both fully open and fully close gate information for gate profiles, hence we have $\underline{MXCNGO = 2}$.

For controlling gates and channel pumps, the time-dependent controlling criteria of upper and lower stage limits for each channel reach should be provided. If each channel reach has a profile different from the others' and 4 data points are needed for the description of both upper and lower stage limits, then we have



Channel Reach Nodes

Channel Element Nodes

Figure 3.2 1-D canal discretization for demonstration

MXCHMXR = 11, MXCHMNR = 11, MXCHMXD = 4, MXCHMND = 4.

Suppose we need 2 time-dependent profiles to give precipitation on channel elements and each profile requires 3 data points to complete the description, we thus need MXCRPR = 2 and MXCRPD = 3. Because there are 2 structure-free joints which are not next to each other, we have 2 groups of channel reaches in which channel reaches are connected by structure-free joints. Therefore, we have MXCNJR = 2. If at most 10000 channel time steps, which can be automatically adjusted in the computer code, are allowed to share 1 subsurface time step, then we have MXCNTC = 10000.

If the second or the third approach is taken in subroutine QSIDE or QSIDE1 to compute the volumetric flow rate through the subsurface-channel interface, we need to specify the interface on the subsurface boundary. Suppose we have 200 subsurface boundary sides and 231 subsurface boundary nodes to describe the interface, we thus need MXCSEL = 200 and MXCSNP = 231.

For the case of South Florida, the water head in the Angel Well was measured in site and the resulting time-dependent profile is to be input if the case is desired. If 20 data points are needed to describe the profile, then we have MXCANG = 20.

For the discretization of overland region, the maximum connecting number to any of the 60 nodes in the region of interest is 9, i.e., MXOJBD = 9 and the maximum number of elements connecting to any node is 4, i.e., MXOKBD = 4. There will be 11 element-sides each on the right and left sides and 4 element-sides each on the northern and southern sides of the region. Thus, there will be a total of 30 element-sides, i.e., MXOBES = 30. Similarly, we can compute the boundary nodes to be 30, i.e., MXOBNP = 30. The half bandwidth plus 1 is equal to 13, i.e., MXOHBP = 13.

To specify maximum control-integers for boundary conditions, on the left side, fluxes from the adjacent aquifer are known. On the right side, the boundary adjacent to the retention pond is impermeable. On the northern and southern sides, prescribed water depth and radiation boundary conditions are

implemented, respectively. There are 11 nodes on the left face and 10 element-sides; thus MXOCNP = 11 and MXOCES = 10. It is further assumed that there two different fluxes going into the region through the left face and each flux can be described by four data points as function of time, i.e., MXOCPR = 2, and MXOCDP = 4. On the northern or southern side, there are 5 nodes and 4 element sides. Since only one prescribed head with 4 data points on the northern side is given, MXODNP = 5, MXODPR = 1, and MXODDP = 4. Also, only one rating curve with 10 data points is prescribed for the southern radiation boundary, therefore, MXONNP=5, MXONES = 4, MXONPR =1, and MXONDP = 10. In this report, we will assume that the whole region of interest is composed of three different kinds of materials. We then have MXOMAT = 3. If we assume that we will make 100 canal time steps in each overland time step, then we have MXOVTK = 100.

For the retention pond, there is one spillway and 1 rating curve with 20 data points to describe the flow rate through this outlet, i.e., MXOULK = 1, MXFXPK = 1, and MXFXDK = 20.

To specify the parameters required for the graphic interface with GMS, there are 12 xy series for describing soil property function, 11 for the sources/sinks and boundary condition of 3-D subsurface, 18 for 1-D canal flow, 2 for the rainfall and rating curve of retention pond, and 7 xy series for the boundary condition of 2-D overland flow. Therefore, MXXYSK = 50. Among these xy series, the maximum no. of data points in a curve is 24, i.e., MXXYPK = 24. In addition, five cross sections, which include 10 nodes in each section, in the overland region have been specified to investigate the hydrograph, which is expected to have 30 points in each time series. Hence, MXPN TK = 10, MXLINK = 5, and MXITPK = 30 are used to store the data for the plotting purpose of hydrograph. Moreover, 10 specific times are specified to store the output results for plotting purpose, i.e., MXPSTK = 10.

Based on the above discussion and the assumed default value, which is 1, for the parameters not mentioned above, the following PARAMETER STATEMENTS can be used to specify the maximum control-

integers in the including file for the problem at hand:

```

C *****
C ===== MAXIMUM DIMENSION BLOCK FOR 3-D SIMULATIONS
C *****
C
C ----- 1. FOR BOTH 3-D FLOW AND 3-D TRANSPORT
C
C   PARAMETER(MXSNP=504, MXSEL=330, MXSBNP=304, MXSBES=302)
C   PARAMETER(MXSTUBS=4*MXSBNP)
C   PARAMETER(MXSJBD=27, MXSKBD=8, MXSNTI=500, MXSDTC=20)
C   PARAMETER(LTMXSNP=126, LMXSNP=42, LMXSBW=17, MXSREGN=12)
C   PARAMETER(MXSMAT=3, MXSSPPM=4, MXSMPPM=9, MXSRMPK=8)
C
C ----- 2. FOR FLOW SOURCE/SINK AND BOUNDARY CONDITIONS
C
C   PARAMETER(MXSSSELF=11, MXSSPRF=3, MXSSDPF=4, MXSWNPF=10, MXSWPRF=5,
C             MXSWDPF=8)
C   PARAMETER(MXSCNPF=72, MXSCESF=55, MXSCPRF=2, MXSCDPF=4)
C   PARAMETER(MXSNNPF=84, MXSNESF=66, MXSNPRF=1, MXSNDPF=2)
C   PARAMETER(MXSVPNPF=1, MXSVESF=1, MXSVPRF=1, MXSVDPF=1)
C   PARAMETER(MXSDNPF=72, MXSDPRF=20, MXSDDPF=8)
C   PARAMETER(MXSRNPF=1, MXSRPRF=1, MXSRDPF=1, MXSRMAF=1, MXSRESF=1)
C
C ----- 3. FOR TRANSPORT SOURCE/SINK AND BOUNDARY CONDITIONS
C
C   PARAMETER(MXSSSELT=1, MXSSPRT=1, MXSSDPT=1, MXSWNPT=1, MXSWPRT=1,
C             MXSWDPT=1)
C   PARAMETER(MXSCNPT=1, MXSCEST=1, MXSCPRT=1, MXSCDPT=1)
C   PARAMETER(MXSNNPT=1, MXSNEST=1, MXSNPRT=1, MXSNDPT=1)
C   PARAMETER(MXSVPNPT=1, MXSVEST=1, MXSVPRT=1, MXSVDPT=1)
C   PARAMETER(MXSDNPT=1, MXSDPRT=1, MXSDDPT=1)
C   PARAMETER(MXSNPW=1, MXSELW=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR 1-D CANAL FLOW
C *****
C
C   PARAMETER(MXCNRN= 12, MXCNR=11)
C   PARAMETER(MXCEL=28, MXCNP=29)
C   PARAMETER(MXCCPM=5)
C   PARAMETER(MXCGPR=4, MXCGPD=10)
C   PARAMETER(MXCWPR=3, MXCWPD=10)
C   PARAMETER(MXCRPR=2, MXCRPD=3)
C   PARAMETER(MXCPPR=5, MXCPPD=4)
C   PARAMETER(MXCUPR=3, MXCUPD=10)
C   PARAMETER(MXCDPR=1, MXCDPD=10)
C   PARAMETER(MXCNTC=10000)
C   PARAMETER(MXCNJR=2, MXCANG=20)
C   PARAMETER(MXCHMXR=11, MXCHMXD=4)
C   PARAMETER(MXCHMNR=11, MXCHMND=4)
C   PARAMETER(MXCSNP=231, MXCSEL=200)
C   PARAMETER(MXCNGO=2)
C

```



```

C *****
C ===== MAXIMUM DIMENSION BLOCK FOR RETENTION POND
C *****
C
  PARAMETER(MXPONK=1,MXPPRK=1,MXDPPK=4,MXPOBK=22,MXOULK=1)
  PARAMETER(MXFXPK=1,MXFXDK=20)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR 2-D OVERLAND FLOW
C *****
C
  PARAMETER(MXOVTK=100,MXONTI=1,MXODTC=1)
  PARAMETER(MXONP=60,MXOEL=44,MXOBPN=30,MXOBES=30)
  PARAMETER(MXOHBP=12,MXOJBD=9,MXOKBD=4)
  PARAMETER(MXORES=MXOEL,MXORPR=3,MXORDP=24)
  PARAMETER(MXODNP=5,MXODPR=1,MXODDP=4)
  PARAMETER(MXOCNP=11,MXOCES=10,MXOCPR=2,MXOCDP=4)
  PARAMETER(MXONNP=5,MXONES=4,MXONPR=1,MXONDP=10)
  PARAMETER(MXOMAT=3)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR GRAPHIC INTERFACE
C *****
C
  PARAMETER(MXXYPK=24,MXXYSK=50)
  PARAMETER(MXPNTK=10,MXLINK=5,MXITPK=30)
  PARAMETER(MXPSTK=10)

```

3.2 Soil Property Function Specifications

Analytical functions are used to describe the relationships of water content, water capacity, and relative hydraulic conductivity with pressure head. Therefore, the user must supply three functions to compute the water content, water capacity, and relative hydraulic conductivity based on the current value of pressure head. The parameters needed to specify the functional form are read and stored in SPP. One example is shown in the subroutine SPROP3 in the source code. In this example, the water content, water capacity, and relative hydraulic conductivity are given by [van Genuchten 1980]:

$$\theta = \theta_r + \frac{\theta_s - \theta_r}{[1 + (\alpha h)^n]^m} \quad (3.1)$$

$$\frac{d\theta}{dh} = \alpha(n-1)[1 - f(\theta)]^m [f(\theta)](\theta_s - \theta_r) \quad (3.2)$$

$$K_r = [(\theta - \theta_r)/(\theta_s - \theta_r)]^{\frac{1}{2}} \{1 - [1 - f(\theta)]^m\}^2 \quad (3.3)$$

in which

$$f(\theta) = [\theta - \theta_r]/[\theta_s - \theta_r]^{1/m} \quad (3.4)$$

and

$$m = 1 - \frac{1}{n} \quad (3.5)$$

To further demonstrate how we should modify the subroutine SPROP3 in the source code to accommodate the material property functions that are different from those given by Eqs. (3.4) through (3.5), let us assume that the following Fermi types of functions are used to represent the unsaturated hydraulic properties [Yeh 1987]:

$$\theta = \theta_r + (\theta_s - \theta_r)/\{1 + \exp[-\alpha(h - h_\theta)]\} \quad (3.6)$$

$$d\theta/dh = \alpha(\theta_s - \theta_r)\exp[-\alpha(h - h_\theta)]/\{1 + \exp[-\alpha(h - h_\theta)]\}^2, \quad (3.7)$$

and

$$\log_{10}(K_r) = \epsilon/\{1 + \exp[-\beta(h - h_k)]\} - \epsilon, \quad (3.8)$$

where θ_s , θ_r , α , and h_θ are the parameters for computing the water content and water capacity; and β , ϵ , and h_k are the parameters for computing the relative hydraulic conductivity. The following segment of subroutine SPROP3 in the source code must be changed, for this example, to the following form for computing the moisture content and water capacity

```
WCR=SPP(1,MTYP,1)
WCS=SPP(2,MTYP,1)
ALPHA=SPP(3,MTYP,1)
HTHETA=SPP(4,MTYP,1)
EPS=SPP(1,MTYP,2)
BETA=SPP(2,MTYP,2)
HSUBK=SPP(3,MTYP,2)
DO 390 KG=1,8
NP=IE(M,KG)
HNP=HKG(KG)
```

```

      HNP=-HNP
C
C ----- SATURATED CONDITION
C
      IF(HNP.LE.0.0) THEN
        TH(KG,M)=WCS
        DTH(KG,M)=0.0D0
        USKFCT=1.0D0
C
      ELSE
C
C ----- UNSATURATED CASE
C
      EXPAH=DEXP(-ALPHA*(HNP-HTHETA))
      TH(KG,M)=WCR+(WCS-WCR)/(1.0D0+EXPAH)
      DTH(KG,M)=ALPHA*(WCS-WCR)*EXPAH/(1.0D0+EXPAH)**2
      AKRLOG=EPS/(1.0D0+DEXP(-BETA*(HNP-HSUBK))) - EPS
      USKFCT=10.0D0**AKRLOG
      ENDIF

```

3.3 Specification of Discharge Rates for Gates and Weirs

Analytical functions can be used to calculate the discharge rate for gates and weirs. In this case, users need to incorporate their formulae, for both non-submerged and submerged flows, into subroutines QGATE within the IF-block of IRATEG = 0 and QWEIR within the IF-block of IRATEW = 0. For example, in subroutine QGATE we have implemented the following formulae for gates with under flow [Kolkman, 1994]

$$q_{gate}^{non-submerged} = C_c a \sqrt{2g(H_{up} - C_c a)} \quad (3.9)$$

$$q_{gate}^{submerged} = C_c a \sqrt{2g(H_{up} - H_{down})} \quad (3.10)$$

where the superscript of discharge q_{gate} is to differentiate non-submerged and submerged flows; C_c is the contraction coefficient; g is gravity; a is the gate opening; H_{up} is the stage of the upstream reach and H_{down} is the stage of the downstream reach to the gate. The volumetric discharge rate is the multiplication of q_{gate} and the gate width. The associated user-supplied statements in subroutine QGATE are in the following.

```

C

```

```

C ***** FOR THE CASE OF IRATEG=0.
C
  IF(IRATEG.EQ.0)THEN
    G=PRMMFG(1,3,ITYP)
    WIDTH=PRMMFG(1,4,ITYP)
C
C ---- FOR THE CASE OF NON-SUBMERGED FLOW.
C
  IF(HDOWN.LT.HGATE)THEN
    CC1=PRMMFG(1,5,ITYP)
    CC2=PRMMFG(1,6,ITYP)
    CC3=PRMMFG(1,7,ITYP)
    HRATIO=(HDOWN-GATEBE)/A
    CC=CC1+CC2*HRATIO+CC3*HRATIO*HRATIO
    QG=WIDTH*CC*A*DSQRT(2.0D0*G*(HUP-CC*A))
C
    HLTUP=HGATE-HDOWN
    HLTDN=HUP-HGATE
C
C ---- FOR THE CASE OF SUBMERGED FLOW.
C
  ELSE
    CC1=PRMSFG(1,5,ITYP)
    CC2=PRMSFG(1,6,ITYP)
    CC3=PRMSFG(1,7,ITYP)
    HRATIO=(HUP-GATEBE)/A
    CC=CC1+CC2*HRATIO+CC3*HRATIO*HRATIO
    QG=WIDTH*CC*A*DSQRT(2.0D0*G*(HUP-HDOWN))
C
    HLTUP=0.5D0*(HUP-HDOWN)
    HLTDN=0.5D0*(HUP-HDOWN)
  ENDIF
C
C ***** FOR THE CASE OF IRATEG=1.
C
  ELSE

```

In the above, HLTUP and HLTDN are used to control the progress of water rise or drop so that we will not miscalculate the discharge rate. In subroutine QWEIR, the following formulae have been installed for rectangular sharp-crested weir [Kolkman, 1994].

$$Q_{\text{weir}}^{\text{non-submerged}} = \frac{2}{3} C_e \sqrt{2g} w_e h_e^{2/3} \quad (3.11)$$

$$Q_{\text{weir}}^{\text{submerged}} = Q_{\text{weir}}^{\text{non-submerged}} \left(1 + \frac{S}{2}\right) \sqrt{1-S} \quad (3.12)$$

where

$$C_e = 0.602 + 0.075 \frac{h}{p} \quad (3.13)$$

$$w_e = w + K_b \quad (3.14)$$

$$h_e = h + K_h \quad (3.15)$$

$$S = \frac{h_2}{h} \quad (3.16)$$

in which Q_{weir} is the volumetric discharge rate through the weir, h is the elevation of the upstream water level above the crest; p is the weir height; w is the weir width; K_b and K_h are correction coefficients for the effects of viscosity and surface tension, respectively; h_2 is the elevation of the downstream water level above the crest.

The following is the user-supplied IF-block implementing Eqs. (3.11) and (3.12) in subroutine QWEIR.

```

C
C ***** FOR THE CASE OF IRATEW=0.
C
  IF(IRATEW.EQ.0)THEN
    W=PRMMFW(1,3,ITYP)
    WWEIR=PRMMFW(1,4,ITYP)
    G=PRMMFW(1,5,ITYP)
    H=HUP-HWEIR
C
    CE=0.602+0.075*H/P
    HE=H+PRMMFW(1,6,ITYP)
    DKB1=PRMMFW(1,7,ITYP)
    DKB2=PRMMFW(1,8,ITYP)
    DKB3=PRMMFW(1,9,ITYP)
    WRATIO=W/WWEIR
    DKB=DKB1+DKB2*WRATIO+DKB3*WRATIO*WRATIO
    WE=W+DKB
C
C ----- FOR THE CASE OF NON-SUBMERGED FLOW.
C
    QW=2.0D0*CE*DSQRT(2.0D0*G)*WE*DSQRT(HE*HE*HE)
C

```

```

C ---- FOR THE CASE OF SUBMERGED FLOW.
C
C   IF(HDOWN.GT.HWEIR)THEN
C     S=(HDOWN-HWEIR)/H
C     QW=QW*(1.0D0+0.5D0*S)*DSQRT(1.0D0-S)
C
C   ENDIF
C
C ***** FOR THE CASE OF IRATEW=1.
C
C   ELSE

```

One should be aware that the time and length units employed for the above analytical functions are second and meter. Consistent units should be taken through simulations.

3.4 Input and Output devices

3.4.1 File Organization

COSFLOW is designed to be operated in batch mode. The input for COSFLOW is organized into a set of input files. The output from COSFLOW is a combination of screen and file output. A summary of the input and output files is shown in Table 3.1 and Table 3.2.

Table 3.1 Input Files

File Name	Description
Super File	Text file containing a list of all of the input and output files used in a COSFLOW simulation.
Geometry File	Text file containing the data describing the finite element mesh, i.e., nodal coordinates and element topology.
Model File	Text containing analysis parameters and options, material properties, boundary conditions, and initial condition options.
Initial Condition Files	Text or binary files containing initial conditions of concentration, head, velocity, moisture content, water depth on the overland.

Flow Files	Text or binary files containing a previously computed flow solution (pressure head and velocity) which are used to define a 3D flow field for a transport only simulation.
------------	--

Table 3.2 Output Files

File Name	Description
Printed Output	Text file containing a summary of output.
Pressure Head	Text or binary file containing the computed pressure heads. This file is used for post-processing or as initial conditions for a subsequent analysis.
Moisture Content	Text or binary file containing the computed moisture content at nodes. This file is used for post-processing.
Velocity in the subsurface	Text or binary file containing the computed Darcian velocities. This file is used for post-processing.
Concentration	Text or binary file containing the computed concentrations. This file is used for post-processing or as initial conditions for a subsequent analysis.
Canal solution	Text files containing the solution of canal water surface, net flow, flow from upstream, flow from downstream, flow from subsurface, flow from pumping, and flow from rainfall.
Water depth on the overland	Text or binary file containing the solution of water depth on the overland.
Water stage on the overland	Text or binary file containing the solution of water stage on the overland.
Velocity on the overland	Text or binary file containing the solution of velocity distribution on the overland.
Infiltration	Text or binary file containing the infiltration rate through the interface between the overland and the subsurface.
Hydrograph	Text file containing the time vs. volumetric flow rate information through each defined cross section.

Water surface elevation	Text file containing the computed result of water surface elevation for retention ponds.
-------------------------	--

3.4.2 Super File

When COSFLOW is launched, the user is prompted for the name of a single input file. This file is called the "super file" and contains a list of all of the appropriated input and output files used in a particular simulation. Grouping the file names in a super file simplifies file management and eliminates the need to type the names of all of the files each time a simulation is performed. The format of the super file is shown in Figure 3.3.

The first record in the file is the file type identifier. Each of the subsequent records represents an input or an output file. The first field in each record is a four character string identifying the type of the file listed in the record. The second field in each record is the name of the corresponding file. The files should be in the same directory as the super file.

Not all of the files shown in Figure 3.3 are required for every simulation. Some of the initial condition files are not required depending on the initial condition options specified in the model file. Also, the user can specify in the model file not to output some of the solution files.

Table 3.3 lists the filenames in super file and the associated conditional variables, logical unit, file type, variables assigned in COSFLOW, and brief descriptions.

COSSUP	/* File type identifier */
GEOM filename	/* Geometry file for 3-D subsurface*/
BCFT filename	/* Model file for 3-D subsurface*/
PRTF filename	/* Printed output file */
ICHD filename	/* Pressure head initial condition file */
ICMC filename	/* Moisture content initial condition file */
ICVL filename	/* 3-D velocity initial condition file */
ICCN filename	/* Concentration initial condition file */
FLVL filename	/* 3-D flow velocity file (for transport only simulation)*/
FLMC filename	/* Moisture content file (for transport only simulation)*/
PSOL filename	/* Pressure head solution file */
MSOL filename	/* Moisture content (nodal) solution file */
VSOL filename	/* 3-D velocity solution file */
CSOL filename	/* Concentration solution file */
FBIN filename	/* Binary storage file saving all output results for flow */
CBIN filename	/* Binary storage file saving all output results for trans. */
MCGS filename	/* Moisture content (Gaussian) file for hot start*/
CANL filename	/* Canal model file */
CANW filename	/* Canal water surface solution file */
CANN filename	/* Canal net flow solution file */
CANU filename	/* Canal flow (from upstream) solution file */
CAND filename	/* Canal flow (from downstream) solution file */
CANS filename	/* Canal flow (from subsurface) solution file */
CANP filename	/* Canal flow (from pumping) solution file */
CANR filename	/* Canal flow (from rainfall) solution file */
GEO2 filename	/* Geometry file for 2-D overland */
BCF2 filename	/* Model file for 2-D overland */
RPON filename	/* Model file for retention pond */
ICWD filename	/* I.C. file containing 2-D water depth information */
HSOL filename	/* Water depth solution file for 2-D overland */
TSOL filename	/* Water stage solution file for 2-D overland */
QSOL filename	/* Velocity solution file for 2-D overland */
ISOL filename	/* Infiltration solution file for 2-D overland */
HYD1 filename	/* The hydrograph file for the 1 st cross section */
HYD2 filename	/* The hydrograph file for the 2 nd cross section */
HYD3 filename	/* The hydrograph file for the 3 rd cross section */
HYD4 filename	/* The hydrograph file for the 4 th cross section */
HYD5 filename	/* The hydrograph file for the 5 th cross section */
RPN1 filename	/* The water surface elevation file for the 1 st retention pond */
RPN2 filename	/* The water surface elevation file for the 2 nd retention pond */
RPN3 filename	/* The water surface elevation file for the 3 rd retention pond */
RPN4 filename	/* The water surface elevation file for the 4 th retention pond */
RPN5 filename	/* The water surface elevation file for the 5 th retention pond */

Figure 3.3 Super File Format

Table 3.3 List of filenames, the associated conditional variables, logical unit, file type, variables assigned in COSFLOW, and brief descriptions

HEC Card	Logical Unit	Conditional Variable	File Type	Variable Assigned	Description
COSSUP	1		ASCII		Super file providing all the filenames required for the simulation
GEOM	10		ASCII		Geometry file for 3-D subsurface
	24		ASCII	FNAME = 3dfemgeo.ech	Echo file for reading 3-D geometry file
BCFT	15		ASCII		Model file for 3-D subsurface
	24		ASCII	FNAME = 3dfemwbc.ech	Echo file for reading 3-D model file
PRTF	16		ASCII		Printed output file
ICHD	31	IFILE=1	Binary	JICH=1, KPH=0	Pressure head initial condition file
	32	IFILE=0	ASCII	JICH=-1, KPH=0	
ICMC	33	IFILE=1	Binary	JICM=1, KMC=0	Moisture content initial condition
	34	IFILE=0	ASCII	JICM=-1, KMC=0	
ICVL	35	IFILE=1	Binary	JICV=1, KVL=0	3-D velocity initial condition file
	36	IFILE=0	ASCII	JICV=-1, KVL=0	
ICCN	37	IFILE=1	Binary	JICC=1, KCN=0	Concentration initial condition file
	38	IFILE=0	ASCII	JICC=-1, KCN=0	
FLVL	51	IVFILE=1	Binary	JIFV=1	3-D flow velocity file (for transport only simulation)
	52	IVFILE=0	ASCII	JIFV=-1	
FLMC	53	IVFILE=1	Binary	JIFM=1	moisture content file (for transport only simulation)

	54	IVFILE=0	ASCII	JIFM=-1	
PSOL	42	IFILE=1	Binary	JPH=1, KPH=1	Pressure head solution file
	46	IFILE=0	ASCII	JPH=-1, KPH=1	
CSOL	43	IFILE=1	Binary	JCN=1, KCN=1	Concentration solution file
	47	IFILE=0	ASCII	JCN=-1, KCN=1	
VSOL	44	IFILE=1	Binary	JVL=1, KVL=1	3-D flow velocity solution file
	48	IFILE=0	ASCII	JVL=-1, KVL=1	
MCGS	41	IFILE=1	Binary	JMC=1	Moisture content (Gaussian) solution file for hot start
	45	IFILE=0	ASCII	JMC=-1	
MSOL	40	IFILE=1	Binary	JMN=1	Moisture content (nodal) solution file
	49	IFILE=0	ASCII	JMN=-1	
CANL	4		ASCII		Canal input file
	24		ASCII	FNAME= canal.ech	Echo file for reading canal input file
CANW	71	IFILE=1	Binary	JCWS=1	Canal water surface solution file
	61	IFILE=0	ASCII	JCWS=-1	
CANN	72	IFILE=1	Binary	JCNE=1	Canal net flow solution file
	62	IFILE=0	ASCII	JCNE=-1	
CANU	73	IFILE=1	Binary	JCUP=1	Canal flow (from upstream) solution file
	63	IFILE=0	ASCII	JCUP=-1	
CAND	74	IFILE=1	Binary	JCDS=1	Canal flow (from downstream) solution file
	64	IFILE=0	ASCII	JCDS=-1	
CANS	75	IFILE=1	Binary	JCSU=1	Canal flow (from subsurface) solution file
	65	IFILE=0	ASCII	JCSU=-1	

CANP	76	IFILE=1	Binary	JCPU=1	Canal flow (from pumping) solution file
	66	IFILE=0	ASCII	JCPU=-1	
CANR	77	IFILE=1	Binary	JCRA=1	Canal flow (from rainfall) solution file
	67	IFILE=0	ASCII	JCRA=-1	
GEO2	10		ASCII		Geometry file for 2-D overland
	24		ASCII	FNAME=2dwmsgeo.ech	Echo file for reading 2-D overland geometry file
BCF2	18		ASCII		Model file for 2-D overland flow
	24		ASCII	FNAME=2dwmswbc.ech	Echo file for reading 2-D model file
RPON	19		ASCII		Model file for retention ponds
	24		ASCII	FNAME=2drponbc.ech	Echo file for reading retention pond model file
FBIN	11		Binary		Binary storage file saving all output results for flow
CBIN	12		Binary		Binary storage file saving all output results for concentration
ICWD	20	IFILE=1	Binary	JICD=1	I. C. file containing 2-D water depth information, FNWD1=FNAME
	21	IFILE=0	ASCII	JICD=-1	
HSOL	22	IFILE=1	Binary	JWD=1	Water depth solution file for 2-D overland flow, FNWD2=FNAME, if FNWD2 is not equal to FNWD1, the KWD=1.
	23	IFILE=0	ASCII	JWD=-1	
TSOL	50	IFILE=1	Binary	JWT=1	Water stage solution file for 2-D overland flow. KWT=1
	60	IFILE=0	ASCII	JWT=-1	
QSOL	26	IFILE=1	Binary	JV2=1	Velocity solution file for 2-D overland flow
	27	IFILE=0	ASCII	JV2=-1	
ISOL	28	IFILE=1	Binary	JIR=1	Infiltration solution file for 2-D overland flow
	29	IFILE=0	ASCII	JIR=-1	
HYD1	55		ASCII		The hydrograph file for the 1 st cross section

HYD2	56		ASCII		The hydrograph file for the 2 nd cross section
HYD3	57		ASCII		The hydrograph file for the 3 rd cross section
HYD4	58		ASCII		The hydrograph file for the 4 th cross section
HYD5	59		ASCII		The hydrograph file for the 5 th cross section
RPN1	5		ASCII		The water surface elevation file for the 1 st retention pond
RPN2	6		ASCII		The water surface elevation file for the 2 nd retention pond
RPN3	7		ASCII		The water surface elevation file for the 3 rd retention pond
RPN4	8		ASCII		The water surface elevation file for the 4 th retention pond
RPN5	9		ASCII		The water surface elevation file for the 5 th retention pond

COSFLOW.DOC - 76

4. EXAMPLE PROBLEMS

Five examples (example one with two cases) are designed to demonstrate the input data and the simulation results of COSFLOW. There are two cases in example one for 2-D overland flow simulation, case A is used to verify the model and case B is a radiation boundary problem. Example two is a 3-D steady-state subsurface flow problem with a pumping well. Example 3, originally designed for 3DFEMFAT [ver. 2C.0, Yeh, 1995], is a coupled 1-D channel water budget and 3-D subsurface flow problem. Example 4 is a 2-D overland flow and 3-D subsurface flow problem with a non-uniform rainfall. Example 5 is a coupled 1-D channel water budget, 2-D overland flow, and 3-D subsurface flow problem with a uniform rainfall and a radiation boundary for 2-D overland domain.

4.1 Example 1A: 2-D Overland Flow

This example is used to verify the simulation of the 2-D overland flow with a sharp geographic jump. The region of interest, as shown in Figure 4.1, is bounded on the left and right by no flux boundary; on the upstream side by Cauchy boundary with a normal flux of $-0.25 \text{ m}^2/\text{sec}$ from $t = 0$ to 5,000 sec; and on the downstream side by Dirichlet boundary with a constant water depth of 0.16 m. The varied bottom slope is designed for creating a sharp jump in the middle of the region as shown in Figure 4.1 and its value is indicated in Figure 4.2. The initial condition is assumed as a uniform water depth of 0.0 m and a Manning coefficient of 0.02 is set through the whole region. For numerical simulation, the domain of interest is discretized into 80 elements and 105 nodes (Figure 4.2). The initial time-step size of 100 sec is used for 100 time steps and then it is changed to 500 sec for another 100 time steps. Therefore, a total 60,000 sec (200 time steps) simulation is made in this example. The water depth tolerance is $2 \times 10^{-4} \text{ m}$ for nonlinear iteration. The relaxation factor for nonlinear iteration is set equal to 1.0 and Direct Gaussian elimination is used to solve

linearized matrix

Upstream B. C.

Cauchy Flux $q = -0.25 \text{ m}^2/\text{sec}$ for $t = 0$ to 5000 sec

$q = 0 \text{ m}^2/\text{sec}$ for $t > 5000$ sec

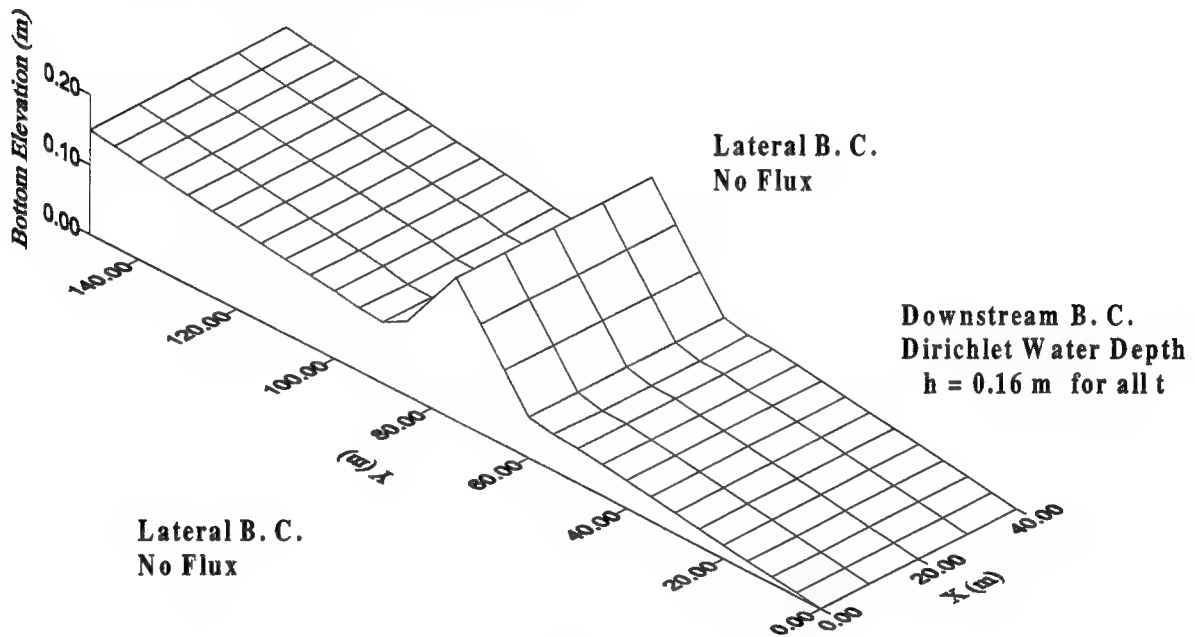
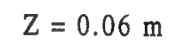


Figure 4.1 Problem definition of Example 1A



79

equations.

To execute the problem, the maximum control-integers in the including file (with extension name .inc)

should be specified as:

```

C
C ----- 1. FOR BOTH 3-D FLOW AND 3-D TRANSPORT
C
  PARAMETER(MXSNP=1,MXSEL=1,MXSBNP=1,MXSBES=1)
  PARAMETER(MXSTUBS=4*MXSBNP)
  PARAMETER(MXSJBD=27,MXSKBD=8,MXSNTI=1,MXSDTC=1)
  PARAMETER(LTMXSNP=1,LMXSNP=1,LMXSBW=1,MXSREGN=1)
  PARAMETER(MXSMAT=1,MXSPPM=1,MXSMPPM=14,MXSRMP=8)
C
C ----- 2. FOR FLOW SOURCE/SINK AND BOUNDARY CONDITIONS
C
  PARAMETER(MXSSELF=1,MXSPPRF=1,MXSDDPF=1,MXSWNPF=1,MXSWPRF=1,
>    MXSWDPF=1)
  PARAMETER(MXSCNPF=1,MXSCESF=1,MXSCPRF=1,MXSCDPF=1)
  PARAMETER(MXSNNPF=1,MXSNEF=1,MXSNPRF=1,MXSNDPF=1)
  PARAMETER(MXSVNPF=1,MXSVEF=1,MXSVPRF=1,MXSVDPF=1)
  PARAMETER(MXSDNPF=1,MXSDPRF=1,MXSDDPF=1)
  PARAMETER(MXSRNPF=1,MXSRPRF=1,MXSRDPF=1,MXSRMAF=1,MXSRESF=1)
C
C ----- 3. FOR TRANSPORT SOURCE/SINK AND BOUNDARY CONDITIONS
C
  PARAMETER(MXSSELT=1,MXSPPRT=1,MXSDDPT=1,MXSWNPT=1,MXSWPRT=1,
>    MXSWDPT=1)
  PARAMETER(MXSCNPT=1,MXSCEST=1,MXSCPRT=1,MXSCDPT=1)
  PARAMETER(MXSNNPT=1,MXSNEST=1,MXSNPRT=1,MXSNDPPT=1)
  PARAMETER(MXSVNPT=1,MXSVEST=1,MXSVPRT=1,MXSVDPT=1)
  PARAMETER(MXSDNPT=1,MXSDPRT=1,MXSDDPT=1)
C
  PARAMETER(MXSNPW=1,MXSELW=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR 1-D CANAL FLOW
C *****
C
  PARAMETER(MXCNRN=1,MXCNR=1)
  PARAMETER(MXCEL=1,MXCNP=1)
  PARAMETER(MXCCPM=1)
  PARAMETER(MXCGPR=1,MXCGPD=1)
  PARAMETER(MXCWPR=1,MXCWPD=1)
  PARAMETER(MXCRPR=1,MXCRPD=1)
  PARAMETER(MXCPPR=1,MXCPPD=1)
  PARAMETER(MXCUPR=1,MXCUPD=1)
  PARAMETER(MXCDPR=1,MXCDPD=1)
  PARAMETER(MXCNTC=1)
  PARAMETER(MXCNJR=1,MXCANG=1)
  PARAMETER(MXCHMXR=1,MXCHMXD=1)
  PARAMETER(MXCHMNR=1,MXCHMND=1)
  PARAMETER(MXCSNP=1,MXCSEL=1)
  PARAMETER(MXCNGO=1)

```

```

C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR RETENTION POND
C *****
C
C   PARAMETER(MXPONK=1,MXPPRK=1,MXDPPK=1,MXPOBK=1,MXOULK=1)
C   PARAMETER(MXFXPK=1,MXFXDK=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR 2-D OVERLAND FLOW
C *****
C
C   PARAMETER(MXOVTK=1,MXONTI=200,MXODTC=4)
C   PARAMETER(MXONP=105,MXOEL=80,MXOBNP=50,MXOBES=50)
C   PARAMETER(MXOHBP=7,MXOJBD=9,MXOKBD=6)
C   PARAMETER(MXORES=MXOEL,MXORPR=1,MXORDP=4)
C   PARAMETER(MXODNP=5,MXODPR=1,MXODDP=2)
C   PARAMETER(MXOCNP=5,MXOCES=4,MXOCPR=1,MXOCDP=4)
C   PARAMETER(MXONNP=5,MXONES=5,MXONPR=3,MXONDP=4)
C   PARAMETER(MXOMAT=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR GRAPHIC INTERFACE
C *****
C
C   PARAMETER(MXXYPK=4,MXXYSK=3)
C   PARAMETER(MXPNTK=10,MXLINK=5,MXITPK=30)
C   PARAMETER(MXPSTK=100)

```

With the above descriptions, the input data can be prepared according to the instructions in Appendix

A. To save space, the input data files and output results are given in the attached floppy disks. Figure 4.3 depicts the water stage distribution at $t = 100$ sec, 5,000 sec and 60,000 sec as simulated by COSFLOW.

As shown in Figure 4.2, the highest elevation (0.21 m) throughout the whole domain is larger than the downstream Dirichlet B.C. (0.16 m). The water supplied by the downstream boundary condition is impossible to cross this jump toward upstream. However, the designed upstream Cauchy flux is capable to provide water for filling up this side of ridge. According to the given geometry (Figure 4.1), the minimum volume of water required to fill the upper area is 258 m^3 , which is obtained based on a water stage of 0.21 m on the upper area. It means it requires 25.8 sec, which is obtained from dividing this amount of water by the volumetric flow rate $10 (= 0.25 \times 40) \text{ m}^3/\text{sec}$, to reach the opposite ridge, i.e., the water starts accumulating on the front area and moves toward downstream after 25.8 sec. Figure 4.3a is the water stage distribution at $t = 100$ sec and shows

satisfactory results with the above estimation.

At $t = 100$ sec, the upstream side of ridge is full of water. When the water stage of the front area is greater than the prescribed boundary condition 0.16 m, the water is turned to flow out of the domain. As shown in Figure 4.2, 80% ($= 120/150 \times 100\%$) of the domain has a bottom slope of 0.001 and the region between $y = 60$ m and $y = 75$ m has the largest slope 0.01. For simplification, we choose 0.01 as the friction slope in estimating the outflow velocity through the downstream boundary by using Manning's formula. Based on (1) the hydraulic radius R is 0.16, which is approximated by the Dirichlet water depth, and (2) the given Manning's n is 0.02, the estimated outflow velocity is $1.47 (= 0.16^{0.6667} \times 0.01^{0.5} / 0.02)$ m/sec, which is approximately equal to the largest simulated velocity (ranging from 0 to 1.425 m/sec) on the downstream boundary. Even with the overestimated volumetric flow rate of $0.24 (= 1.47 \times 0.16)$ m³/sec per unit length, the upstream inflow rate 0.25 m³/sec per unit length is still greater than this outflow rate. That means the water will gradually accumulate in the domain. Because the upstream Cauchy flux constantly supplies water into the domain from $t = 0$ sec to $t = 5,000$ sec and the downstream boundary condition is always held at 0.16 m, the maximum water stage will occur at $t = 5,000$ sec. According to the simulated results at $t = 5,000$ sec, no significant amount of water ($= 1.7 \times 10^{-13}$ m³/sec obtained from simulation) increases within the domain. It means the net flow rate ($= 10$ m³/sec) into the domain through the upstream Cauchy boundary should be equal to the net flow rate out of the domain through the downstream Dirichlet boundary ($= 9.94$ m³/sec). The error 0.6% is due to the discretization of space and time for the numerical simulation.

After the recharge from the upstream Cauchy flux stops at $t = 5,000$ sec, the water continuously flows out through the downstream boundary as long as the water stage of the front area is higher than 0.16 m. When the water stage of the upper area is as low as 0.21 m (the highest elevation in the domain), the water stops crossing over the geographical jump to downstream and preserves in the upper area without any source/sink. Thus, the water stage at the last time step ($t = 60,000$ sec) is expected to be 0.21 m within the upper area. As

to the front area, the water stage should be controlled to 0.16 m by the prescribed Dirichlet boundary value.

Evidence in Figure 4.3c shows that the above statements are correct.

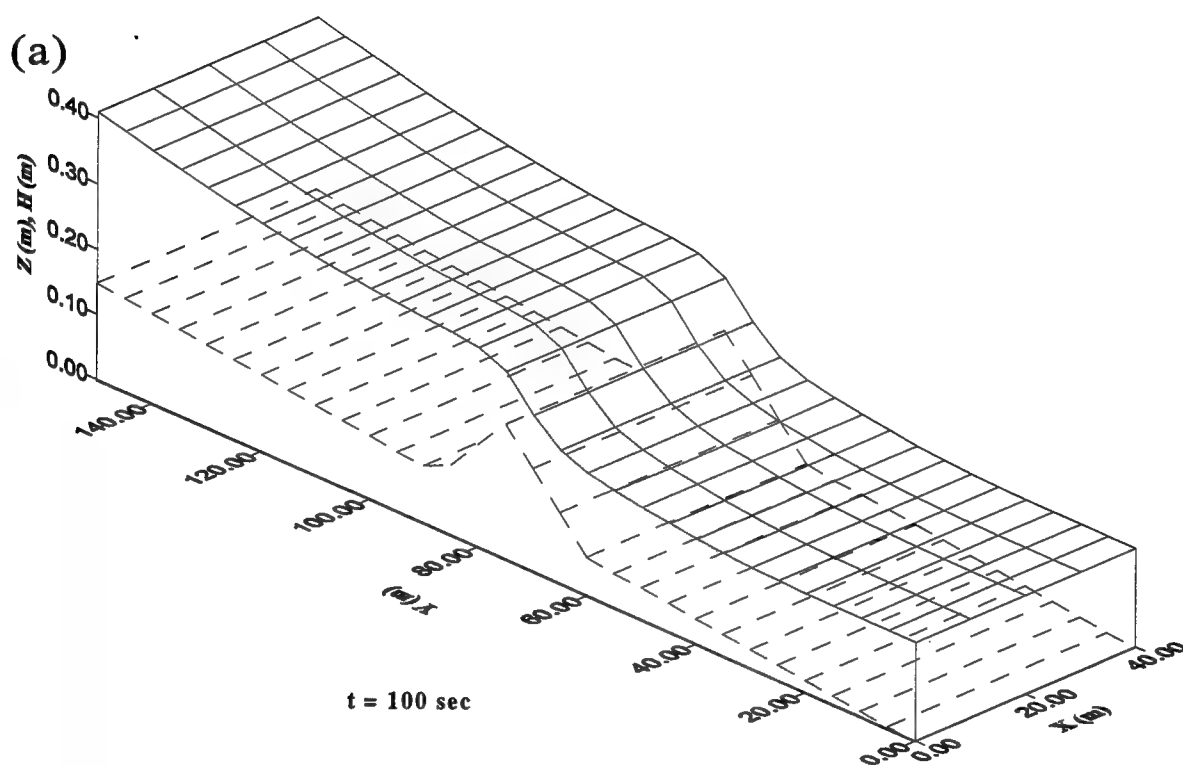


Figure 4.3a Simulated water stage at $t = 100$ sec for Example 1A

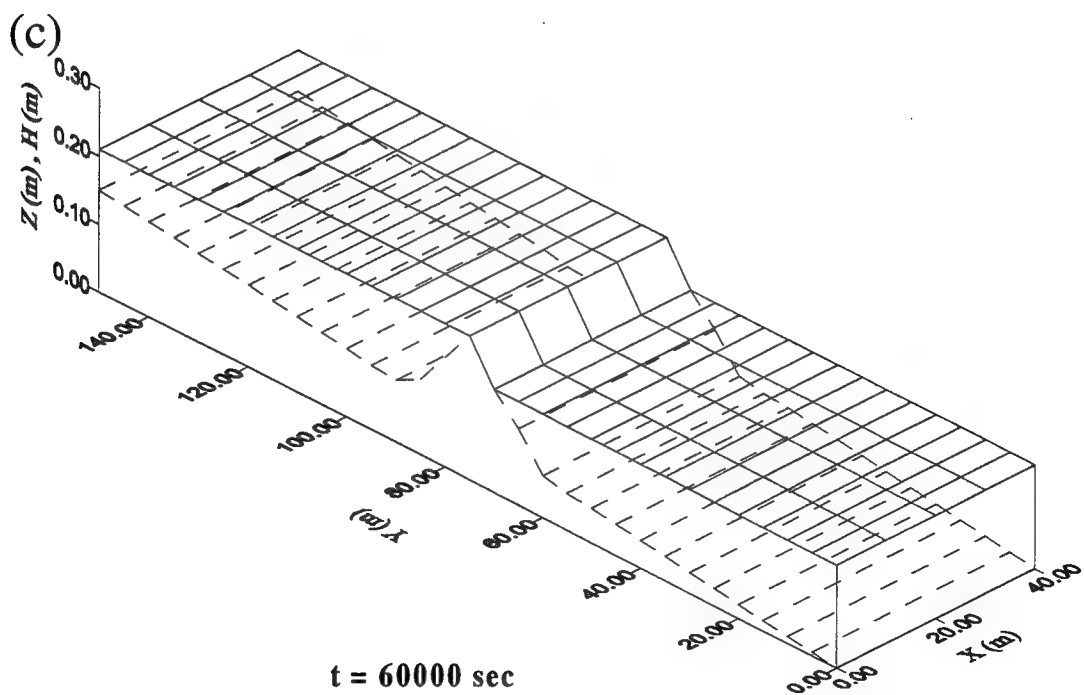
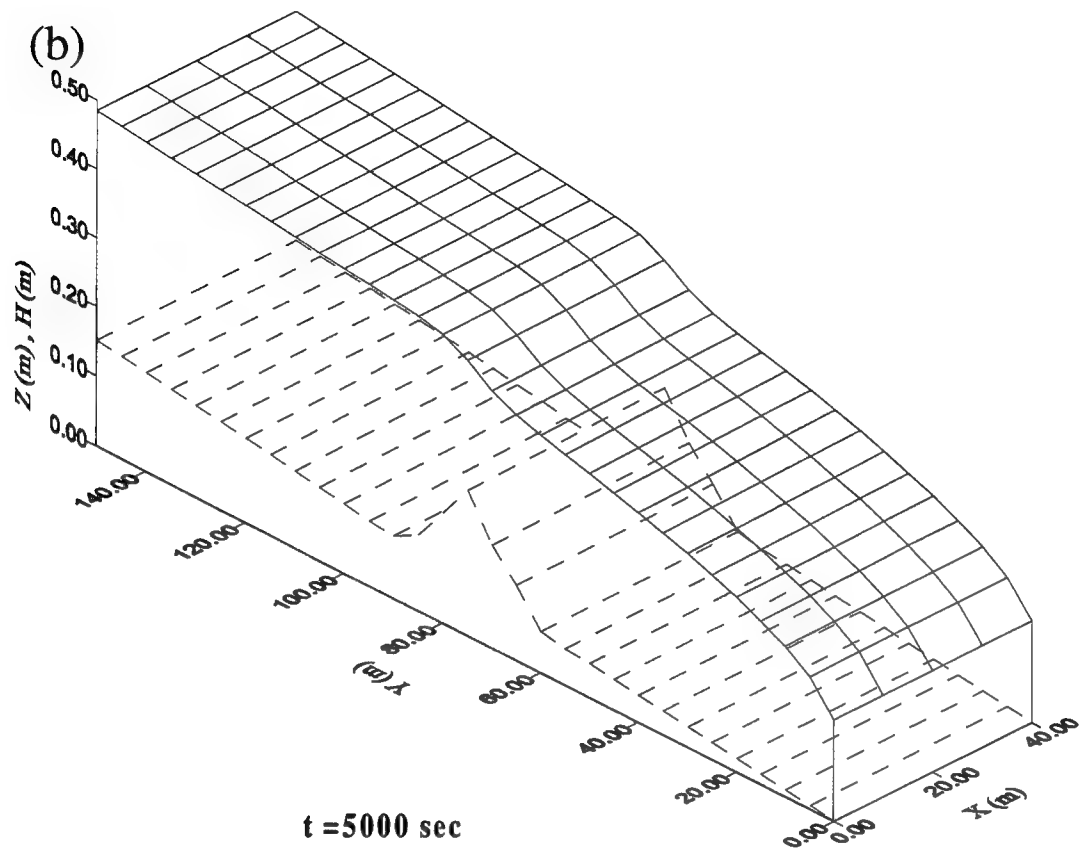


Figure 4.3 Simulated water stage at (b) $t = 5,000 \text{ sec}$ and (c) $t = 60,000 \text{ sec}$ for Example 1A

4.2 Example 1B: 2-D Overland Flow

This example is used to demonstrate the simulation of the 2-D overland flow with a radiation boundary and a non-uniform rainfall. The same region of interest as Example 1A, is now bounded on the upstream (i.e., $y = 150$ m), left (i.e., $x = 0$ m), and right (i.e., $x = 40$ m) sides by no flux boundary; on the downstream (i.e., $y = 0$ m) side with a radiation boundary from $x = 10$ m to 30 m and no flux from $x = 0$ m to 10 m and $x = 30$ m to 40 m; and having a rainfall of 5×10^{-5} m/sec on $y = 80$ m to 150 m from $t = 0$ sec to 5000 sec (Figure 4.4). The rating curve for the radiation boundary is given in Figure 4.5. The bottom slope is the same as Example 1A (Figure 4.2). The initial condition is assumed as a uniform water depth of 0.0 m and a Manning coefficient of 0.02 is set through the whole region. For numerical simulation, the domain of interest is discretized into 80 elements and 105 nodes (Figure 4.2). A constant time step size of 50 sec is used for 200 time steps. The water depth tolerance is 2×10^{-5} m for nonlinear iteration. The relaxation factor for nonlinear iteration is set equal to 1.0 and Direct Gaussian elimination is used to solve linearized matrix equations.

To execute the problem, the maximum control-integers in the including file should be specified as:

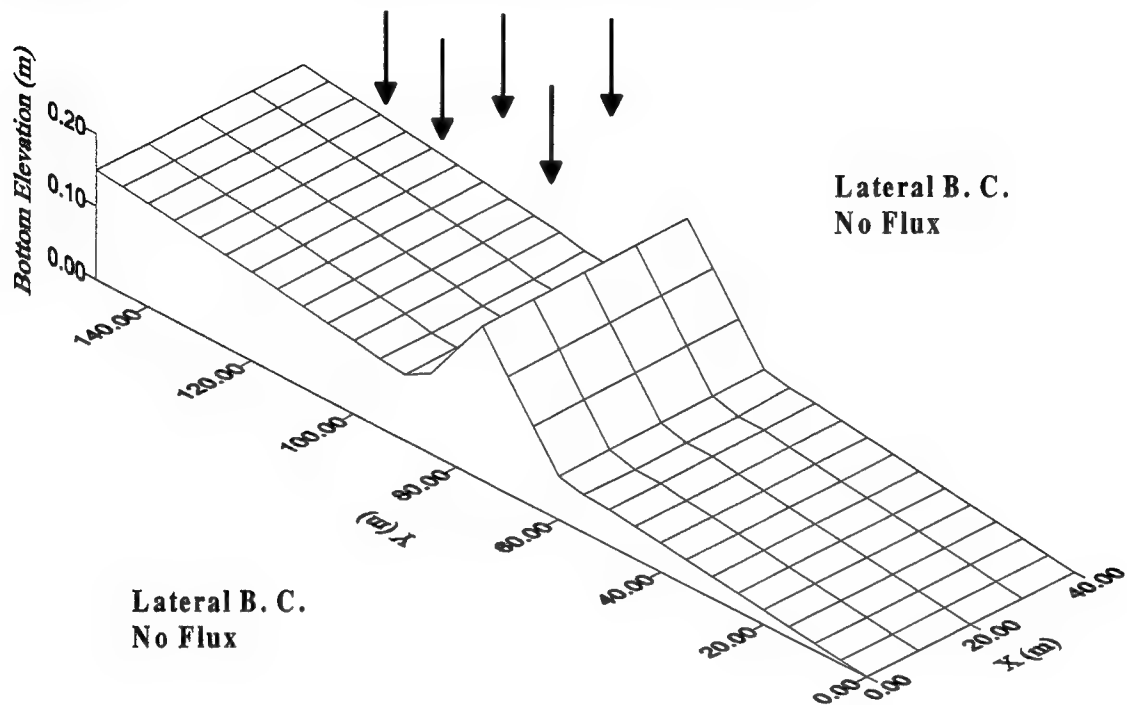
```

C
C ----- 1. FOR BOTH 3-D FLOW AND 3-D TRANSPORT
C
  PARAMETER(MXSNP=1,MXSEL=1,MXSBNP=1,MXSBES=1)
  PARAMETER(MXSTUBS=4*MXSBNP)
  PARAMETER(MXSJBD=27,MXSKBD=8,MXSNTI=1,MXSDTC=1)
  PARAMETER(LTMXSNP=1,LMXSNP=1,LMXSBW=1,MXSREGN=1)
  PARAMETER(MXSMAT=1,MXSPPM=1,MXSMPPM=14,MXSRMP=8)
C
C ----- 2. FOR FLOW SOURCE/SINK AND BOUNDARY CONDITIONS
C
  PARAMETER(MXSSELF=1,MXSPPRF=1,MXSDDPF=1,MXSWNPF=1,MXSWPRF=1,
>    MXSWDPF=1)
  PARAMETER(MXSCNPF=1,MXSCESF=1,MXSCPRF=1,MXSCDPF=1)
  PARAMETER(MXSNNPF=1,MXSNEF=1,MXSNPRF=1,MXSNDPF=1)
  PARAMETER(MXSVPNF=1,MXSVEF=1,MXSVPRF=1,MXSVDPF=1)
  PARAMETER(MXSDNPF=1,MXSDPRF=1,MXSDDPF=1)
  PARAMETER(MXSRNPF=1,MXSRPRF=1,MXSRDPF=1,MXSRMAF=1,MXSRESF=1)
C
C ----- 3. FOR TRANSPORT SOURCE/SINK AND BOUNDARY CONDITIONS
C

```

Upstream B. C.
No Flux

For $Y = 80$ to 150 m
 $I = 5 \times 10^{-5}$ m/sec for $t = 0$ to 5000 sec



Lateral B. C.
No Flux

Lateral B. C.
No Flux

Downstream B. C.
Radiation Flux for $X = 10$ to 30 m
No Flux for $X = 0$ to 10 & $X = 30$ to 40 m

Figure 4.4 Problem definition for Example 1B

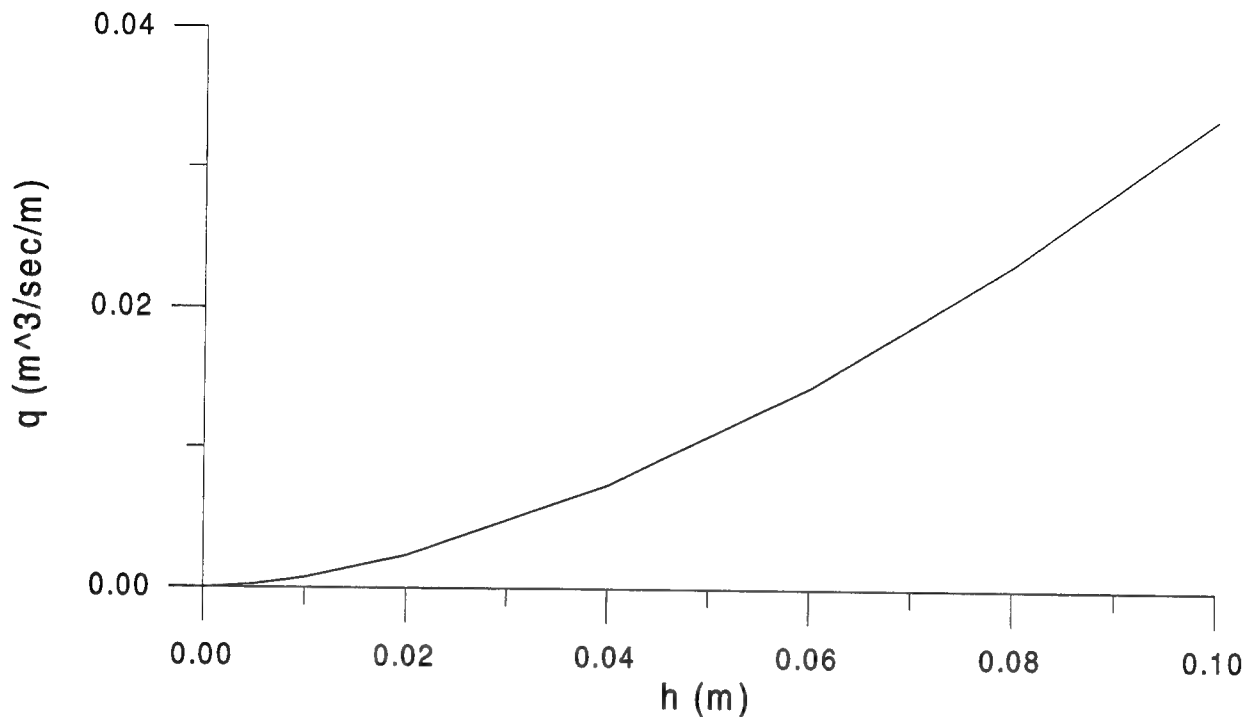


Figure 4.5 The rating curve for the radiation boundary in Example 1B

```

PARAMETER(MXSSELT=1,MXSSPRT=1,MXSSDPT=1,MXSWNPT=1,MXSWPRT=1,
>  MXSWDPT=1)
PARAMETER(MXSCNPT=1,MXSCEST=1,MXSCPRT=1,MXSCDPT=1)
PARAMETER(MXSNNPT=1,MXSNEST=1,MXSNPRT=1,MXSNDPT=1)
PARAMETER(MXS VNPT=1,MXSVEST=1,MXSV PRT=1,MXSVDPT=1)
PARAMETER(MXSDNPT=1,MXSDPRT=1,MXSDDPT=1)
C
PARAMETER(MXSNPW=1,MXSELW=1)
C *****
C === MAXIMUM DIMENSION BLOCK FOR 1-D CANAL FLOW
C *****
C
PARAMETER(MXCNRN=1,MXCNR=1)
PARAMETER(MXCEL=1,MXCNP=1)
PARAMETER(MXCCPM=1)
PARAMETER(MXCGPR=1,MXCGPD=1)
PARAMETER(MXCWPR=1,MXCWPD=1)
PARAMETER(MXCRPR=1,MXCRPD=1)
PARAMETER(MXCPPR=1,MXCPPD=1)
PARAMETER(MXCUPR=1,MXCUPD=1)
PARAMETER(MXCDPR=1,MXCDPD=1)
PARAMETER(MXCNTC=1)
PARAMETER(MXCNJR=1,MXCANG=1)
PARAMETER(MXCHMXR=1,MXCHMXD=1)
PARAMETER(MXCHMNR=1,MXCHMND=1)

```

```

PARAMETER(MXCSPN=1,MXCSEL=1)
PARAMETER(MXCNGO=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR RETENTION POND
C *****
C
PARAMETER(MXPONK=1,MXPPRK=1,MXDPPK=1,MXPOBK=1,MXOULK=1)
PARAMETER(MXFXPK=1,MXFXDK=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR 2-D OVERLAND FLOW
C *****
C
PARAMETER(MXOVTK=1,MXONTI=200,MXODTC=4)
PARAMETER(MXONP=105,MXOEL=80,MXOBPN=50,MXOBES=50)
PARAMETER(MXOHBP=7,MXOJBD=9,MXOKBD=6)
PARAMETER(MXORES=MXOEL,MXORPR=2,MXORDP=4)
PARAMETER(MXODNP=5,MXODPR=1,MXODDP=2)
PARAMETER(MXOCNP=5,MXOCES=4,MXOCPR=1,MXOCDP=4)
PARAMETER(MXONNP=5,MXONES=5,MXONPR=3,MXONDP=15)
PARAMETER(MXOMAT=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR GRAPHIC INTERFACE
C *****
C
PARAMETER(MXXYPK=15,MXXYSK=6)
PARAMETER(MXPNTK=10,MXLINK=5,MXITPK=30)
PARAMETER(MXPSTK=100)

```

With the above descriptions, the input data can be prepared according to the instructions in Appendix

A. To save space, the input data files and output results are given in the attached floppy disks. Figure 4.6 depicts the water stage distributions at $t = 1,850$ sec, $5,000$ sec, and $10,000$ sec as simulated by COSFLOW.

In this example, rainfall on the upper half area supplies water to 2-D overland domain and the downstream radiation boundary serves as an outlet to discharge water according to a given rating curve (Figure 4.5). Same as that calculated in Example 1A, the minimum volume of water required to reach the highest point is 258 m^3 (a water stage of 0.21 m on the upper area). Divided by the volumetric rainfall rate $0.14 (= 5 \times 10^{-5} \times 70 \times 40) \text{ m}^3/\text{sec}$, this amount of water needs $1,842.9$ sec for accumulation during the rainfall period. After that, the water flows over the peak of ridge to the front area of the domain. As we expected, the upper area is full of water, which is shown in Figure 4.6a for the water stage distribution at $t = 1,850$ sec.

At $t = 1,850$ sec, the upstream side of ridge is filled with water and the water on the upper half domain starts to flow toward downstream. As the water stage of the front area is higher than 0.0 m, the discharge rate is gradually increased according to a given rating curve (Figure 4.5). At $t = 5,000$ sec, the simulated water stage on the downstream boundary is less than 0.03 m and the volumetric outflow rate is $0.1 (= 5 \times 10^{-3} \text{ (from Figure 4.5)} \times 20) \text{ m}^3/\text{sec}$. This is an overestimated value because the true water stage is less than 0.03 m on the boundary. According to the problem description, the volumetric rainfall rate is $0.14 (= 5 \times 10^{-5} \times 40 \times 70) \text{ m}^3/\text{sec}$, which is larger than the estimated volumetric outflow rate described above. Therefore, the rainfall is capable to supply water not only for the accumulation on the upper area but also for flowing to the front area of the domain. The highest water stage is expected to occur at $t = 5,000$ sec as shown in Figure 4.6b.

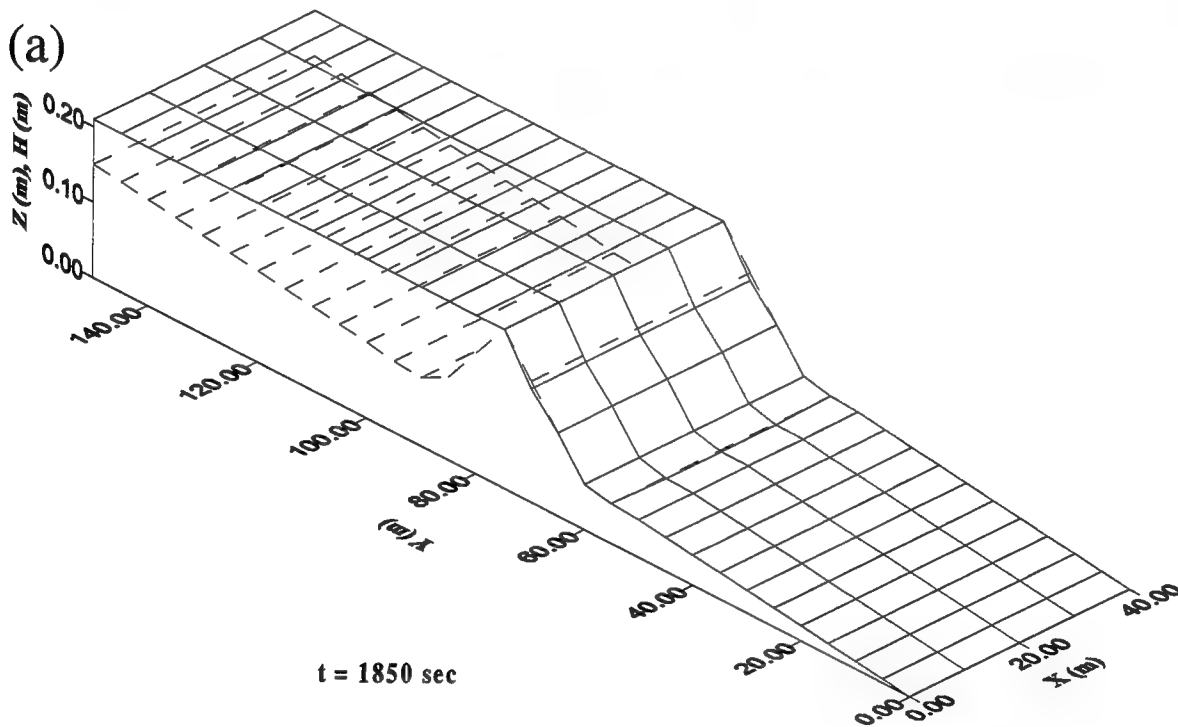


Figure 4.6a Simulated water stage at $t = 1,850$ sec for Example 1B

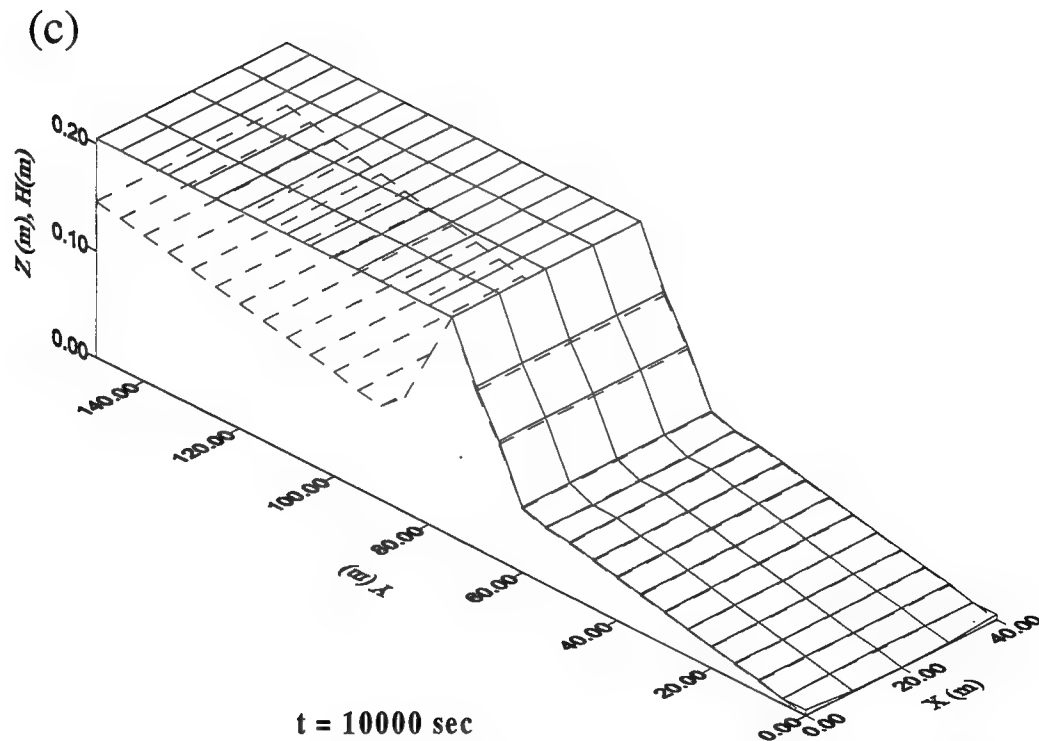
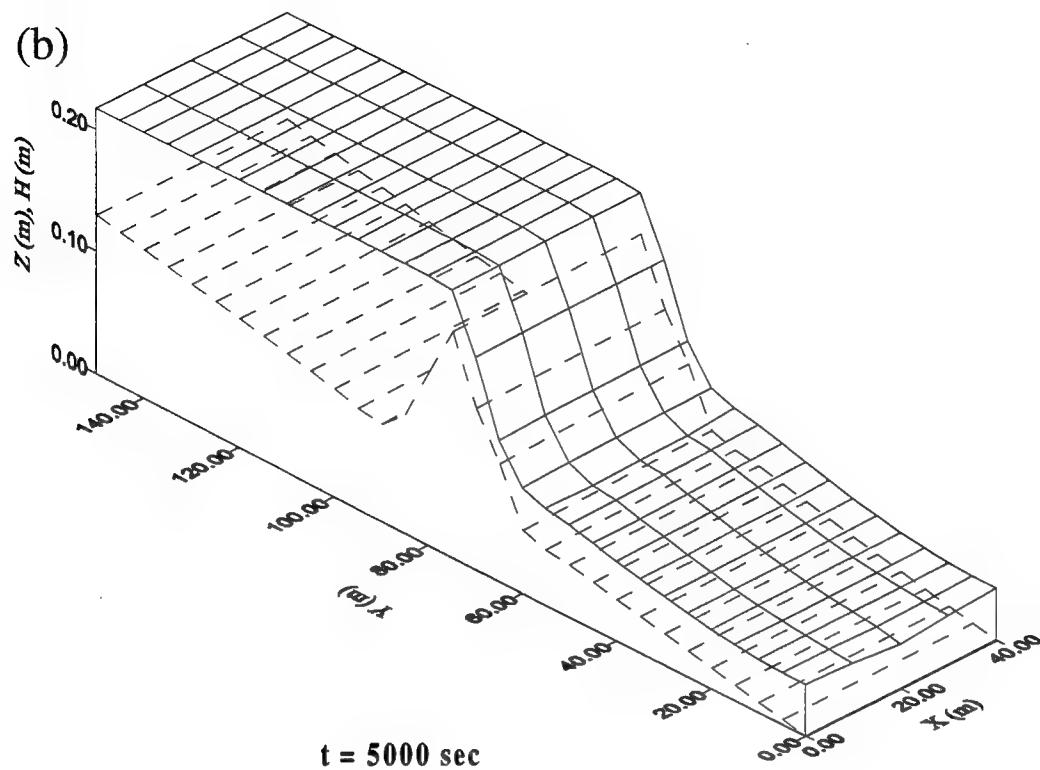


Figure 4.6 Simulated water stage at (b) $t = 5,000 \text{ sec}$ and (c) $t = 10,000 \text{ sec}$ of Example 1B

After the rainfall stops at $t = 5,000$ sec, the water continuously flows out through the downstream boundary as long as the nearby water stage is higher than 0.0 m. When the water stage of the upper area is as low as 0.21 m (the elevation of the geographic jump), the separation of the water occurs. At the last time step ($t = 10,000$ sec), most of water is preserved on the upper area, but it is almost dry in the downstream area due to water flowing out through the radiation boundary. Figure 4.6c shows the above phenomena, i.e., the water stage on the upper area is 0.21 m and most of the downstream area is dry.

4.3 Example 2: 3-D Subsurface Flow

This example is selected to represent the simulation of a three-dimensional problem with COSFLOW. The problem involves the steady state flow to a pumping well. The region of interest is bounded on the left and right by hydraulically connected rivers; on the front, back, and bottom by impervious aquifuges; and on the top by an air-soil interface (Figure 4.7). A pumping well is located at $(x, y) = (540, 400)$ (Figure 4.7). Initially, the water table is assumed to be horizontal and is 60 m above the bottom of the aquifer. The water level at the well is then lowered to a height of 30 m. This height is held until a steady state condition is reached. The medium in the region is assumed to be anisotropic and have saturated hydraulic conductivity components $K_{xx} = 5$ m/d, $K_{yy} = 0.5$ m/d, and $K_{zz} = 2$ m/d. The porosity of the medium θ_s is 0.25 and the field capacity θ_r is 0.0125. The unsaturated characteristic hydraulic properties of the medium are given as

$$\theta = \theta_r + \frac{\theta_s - \theta_r}{1 + (\alpha |h_a - h|)^\beta} \quad (4.1)$$

and

$$K_r = \left[\frac{\theta - \theta_r}{\theta_s - \theta_r} \right]^2 \quad (4.2)$$

where h_a , α , and β are the parameters used to compute the water content and the relative hydraulic conductivity.

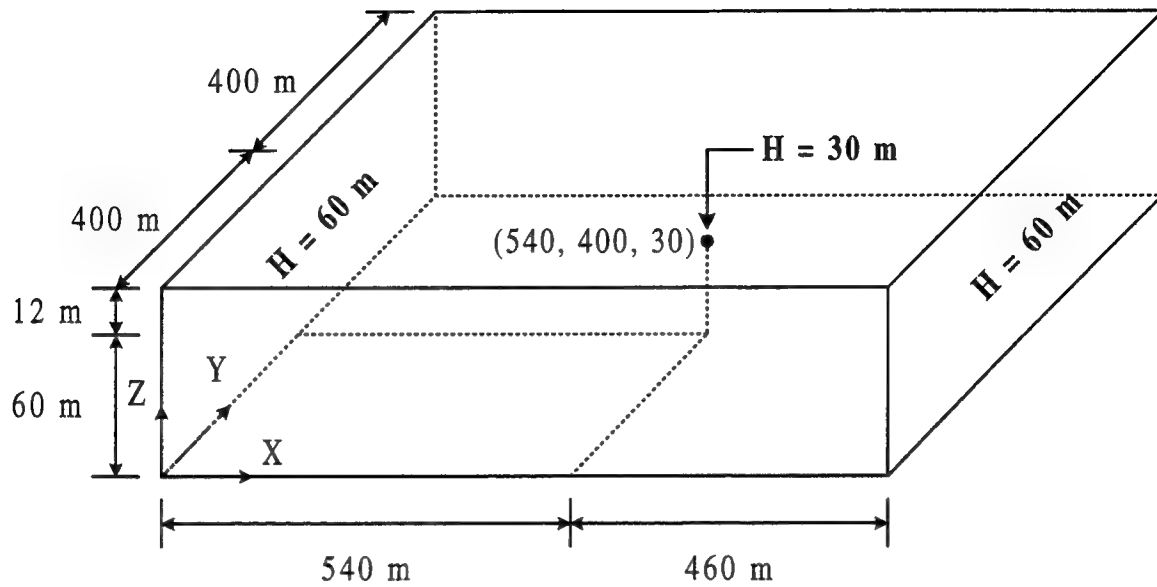


Figure 4.7 Problem definition and sketch for Example 2

Because of the symmetry, the region for numerical simulation will be taken as $0 < x < 1000$ m, $0 < y < 400$ m, and $0 < z < 72$ m. The boundary conditions are given as: pressure head is assumed hydrostatic on two vertical planes located at $x = 0$ and $0 < z < 60$, and $x = 1000$ and $0 < z < 60$, respectively; no flux is imposed on all other boundaries of the flow regime. A steady state solution will be sought. A pre-initial condition is set as $h = 60 - z$.

The region of interest is discretized with $20 \times 8 \times 10 = 1600$ elements resulting in $21 \times 9 \times 11 = 2079$ node points (Figure 4.8). The nodes are located at $x = 0, 70, 120, 160, 200, 275, 350, 400, 450, 500, 540, 570, 600, 650, 750, 800, 850, 900, 950$, and 1000 in the x -direction, and at $z = 0, 15, 30, 35, 40, 45, 50, 55, 60, 66$, and 72 m in the z -direction as reported by Huyakorn et al. (1986). In the y -direction, nodes are spaced evenly at $\Delta z = 50$ m. The solver used to solving the linearized matrix is pointwise iterative successive system.

The pressure head tolerance is 10^{-2} m for nonlinear iteration and is $5 \cdot 10^{-3}$ m for matrix solver. The relaxation factors for nonlinear iteration and matrix solver iteration are set equal to 1.0 and 1.5, respectively.

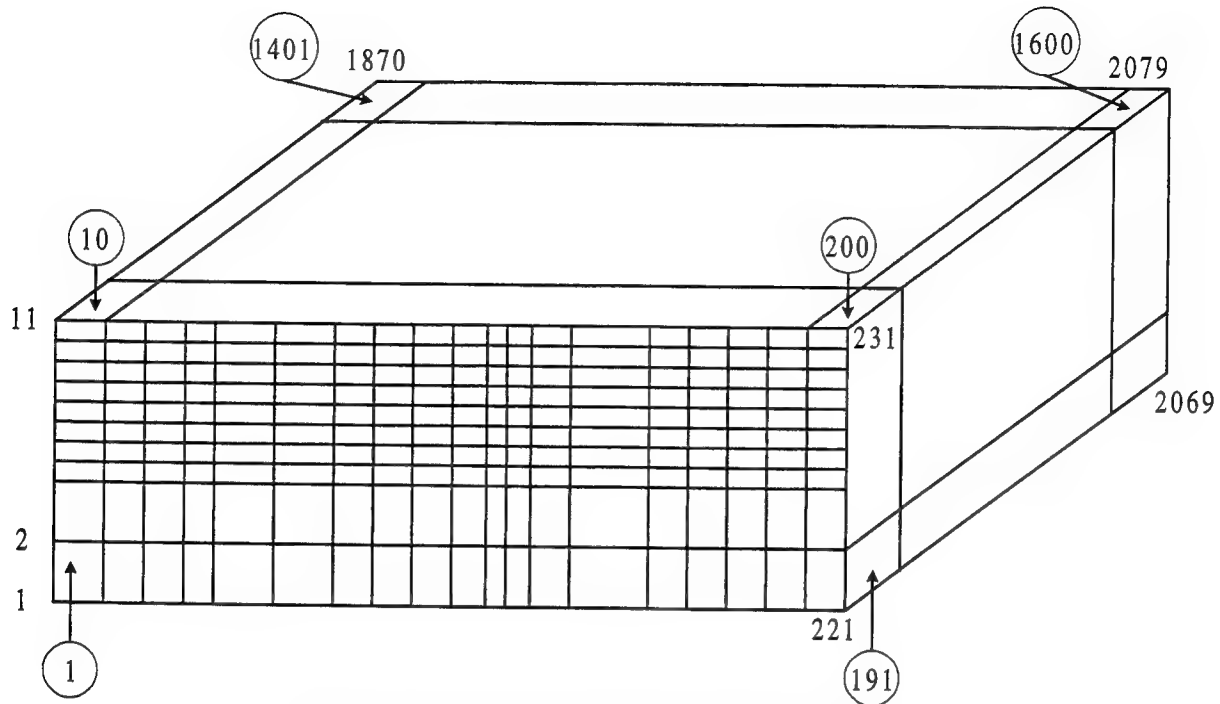


Figure 4.8 Finite element discretization for Example 2

To execute the problem, the maximum control-integers in the MAIN should be specified as

```

C
C ----- 1. FOR BOTH 3-D FLOW AND 3-D TRANSPORT
C
  PARAMETER(MXSNP=2079,MXSEL=1600,MXSBNP=882,MXSBES=880)
  PARAMETER(MXSTUBS=4*MXSBNP)
  PARAMETER(MXSJBD=27,MXSKBD=8,MXSNTI=1,MXSDTC=1)
  PARAMETER(LTMXSNP=1,LMXSNP=1,LMXSBW=1,MXSREGN=1)
  PARAMETER(MXSMAT=1,MXSPPM=5,MXSMPPM=14,MXSRMP=8)
C
C ----- 2. FOR FLOW SOURCE/SINK AND BOUNDARY CONDITIONS
C
  PARAMETER(MXSSELF=1,MXSSPRF=1,MXSSDPF=1,MXSWNPF=1,MXSWPRF=1,
>    MXSWDPF=1)
  PARAMETER(MXSCNPF=1,MXSCESF=1,MXSCPRF=1,MXSCDPF=1)
  PARAMETER(MXSNNPF=1,MXSNESEF=1,MXSNPRF=1,MXSNDPF=1)
  PARAMETER(MXSVNPF=1,MXSVSEF=1,MXSVPRF=1,MXSVDPF=1)
  PARAMETER(MXSDNPF=165,MXSDPRF=2,MXSDDPF=2)
  PARAMETER(MXSRNPF=1,MXSRPRF=1,MXSRDPF=1,MXSRMAF=1,MXSRESF=1)
C
C ----- 3. FOR TRANSPORT SOURCE/SINK AND BOUNDARY CONDITIONS
C
  PARAMETER(MXSSELT=1,MXSSPRT=1,MXSSDPT=1,MXSWNPT=1,MXSWPRT=1,
>    MXSWDPT=1)
  PARAMETER(MXSCNPT=1,MXSCEST=1,MXSCPRT=1,MXSCDPT=1)
  PARAMETER(MXSNNPT=1,MXSNEST=1,MXSNPRT=1,MXSNPT=1)
  PARAMETER(MXSVNPT=1,MXSVEST=1,MXSVPRT=1,MXSVPT=1)
  PARAMETER(MXSDNPT=1,MXSDPRT=1,MXSDPT=1)
C
  PARAMETER(MXSNPW=1,MXSELW=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR 1-D CANAL FLOW
C *****
C
  PARAMETER(MXCNRN=1,MXCNR=1)
  PARAMETER(MXCEL=1,MXCNP=1)
  PARAMETER(MXCCPM=1)
  PARAMETER(MXCGRP=1,MXCGPD=1)
  PARAMETER(MXCWPR=1,MXCWPD=1)
  PARAMETER(MXCRPR=1,MXCRPD=1)
  PARAMETER(MXCPRP=1,MXCPRD=1)
  PARAMETER(MXCUPR=1,MXCUPD=1)
  PARAMETER(MXCDPR=1,MXCDPD=1)
  PARAMETER(MXCNTC=1)
  PARAMETER(MXCNRJ=1,MXCANG=1)
  PARAMETER(MXCHMXR=1,MXCHMXD=1)
  PARAMETER(MXCHMNR=1,MXCHMND=1)
  PARAMETER(MXCSNP=1,MXCSEL=1)
  PARAMETER(MXCNGO=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR RETENTION POND
C *****
C

```



```

PARAMETER(MXPONK=1,MXPPRK=1,MXDPPK=1,MXPOBK=1,MXOULK=1)
PARAMETER(MXFXPK=1,MXFXDK=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR 2-D OVERLAND FLOW
C *****
C
PARAMETER(MXOVTK=1,MXONTI=1,MXODTC=1)
PARAMETER(MXONP=1,MXOEL=1,MXOBNP=1,MXOBES=1)
PARAMETER(MXOHBP=1,MXOJBD=1,MXOKBD=1)
PARAMETER(MXORES=MXOEL,MXORPR=1,MXORDP=1)
PARAMETER(MXODNP=1,MXODPR=1,MXODDP=1)
PARAMETER(MXOCNP=1,MXOCES=1,MXOCPR=1,MXOCDP=1)
PARAMETER(MXONNP=1,MXONES=1,MXONPR=1,MXONDP=1)
PARAMETER(MXOMAT=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR GRAPHIC INTERFACE
C *****
C
PARAMETER(MXXYPK=5,MXXYSK=3)
PARAMETER(MXPNTK=10,MXLINK=5,MXITPK=30)
PARAMETER(MXPSTK=100)

```

To reflect the soil property function given by Eqs. (4.1) and (4.2), we have to modify the subroutine SPROP3 in the source code as follows.

```

WCR=THPROP(1,MTYP)
WCS=THPROP(2,MTYP)
HAA=THPROP(3,MTYP)
ALPHA=THPROP(4,MTYP)
BETA=THPROP(5,MTYP)
DO 800 KG=1,8
HNP=HKG(KG)
HNP=-HNP
C
C ----- SATURATED CONDITION
C
IF(HNP.GT.0) GO TO 700
TH(KG,M)=WCS
AKR(KG,M)=1.0D0
DTH(KG,M)=0.0D0
GO TO 800
C
C ----- UNSATURATED CASE
C
700 THMKG=WCR+(WCS-WCR)/(1.0D0+(ALPHA*DABS(-HNP-HAA))**BETA)

```

```

TH(KG,M)=THMKG
AKR(KG,M)=((THMKG-WCR)/(WCS-WCR))**2
DNOM=1.0D0+(ALPHA*DABS(-HNP-HAA))**BETA
DTH(KG,M)=(WCS-WCR)*(ALPHA*DABS(-HNP-THAA))**((BETA-1.0D0)/DNOM)**2
    
```

With the above descriptions, the input data can be prepared according to the instructions in Appendix A. To save space, the input data files and output results are given in the attached floppy. Figure 4.9 and Figure 4.10 depict the water table distribution and the velocity field along the x-z cross-section through the well as simulated by COSFLOW.

The simulated results agree with those from 3DFEMWATER. Figure 4.9a shows the water table elevation which is gradually reduced from 60 m on the boundaries to 30 m at the well location. The cone of depression is emphasized on the longitudinal section through the pumping well (Figure 4.9b). Figure 4.10a and Figure 4.10b show the velocity field in the domain and on the x-z cross section through the pumping well, respectively. According to the results, the pumping increases the area of unsaturated zone. This is because the ability to conduct water in the unsaturated zone is decreased due to the reduced hydraulic conductivity from Eq. (4.2). Therefore, the inclusion of the unsaturated zone in groundwater flow simulation is important. The neglect of the unsaturated zone in a three-dimensional unconfined aquifer would miscalculate not only the real head distribution but also the velocity field. As a consequence, any simulation of contaminant transport using such an incorrect flow field will be inadequate.

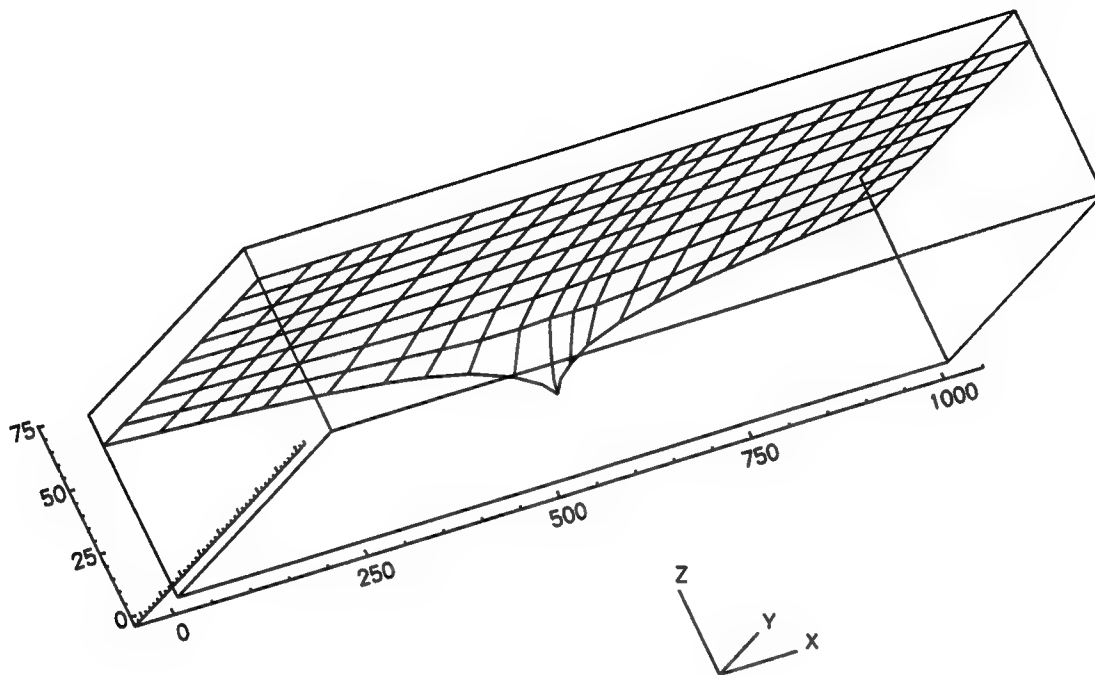


Figure 4.9a Water table for Example 2

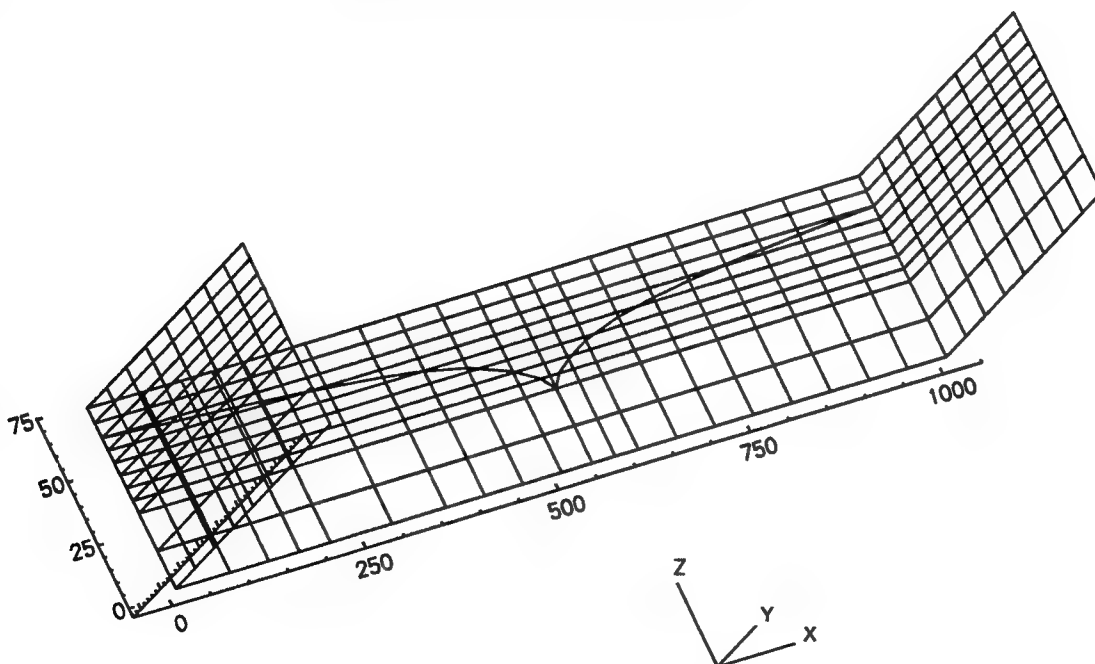


Figure 4.9b Water table on the x-z cross section through the pumping well for Example 2

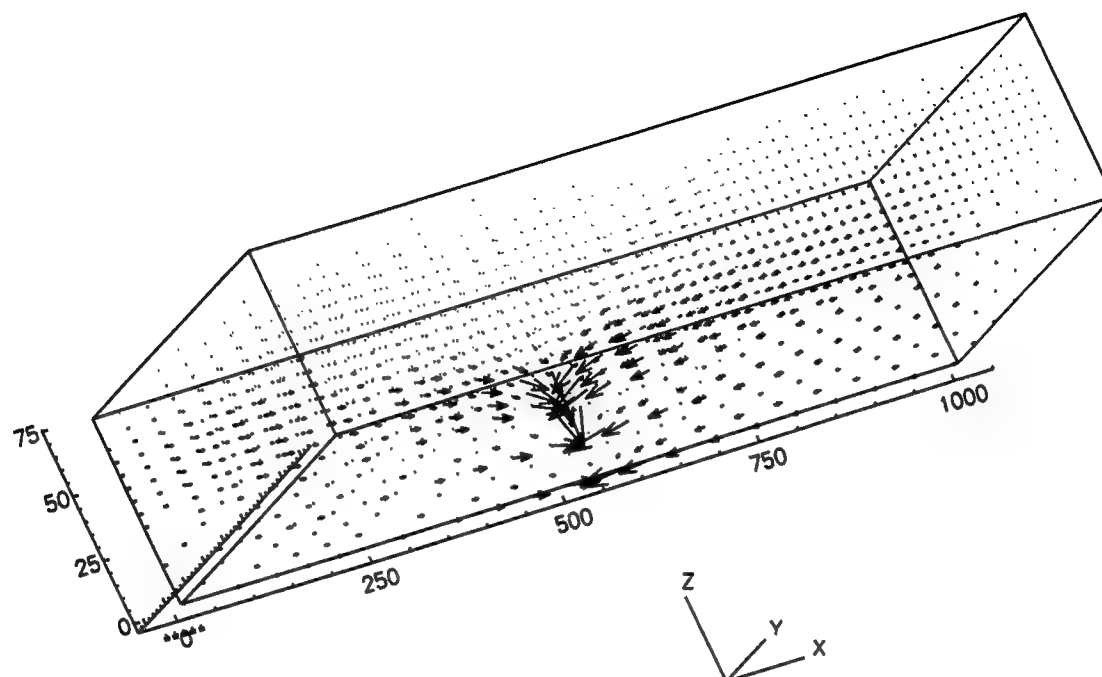


Figure 4.10a Velocity distribution throughout the domain for Example 2

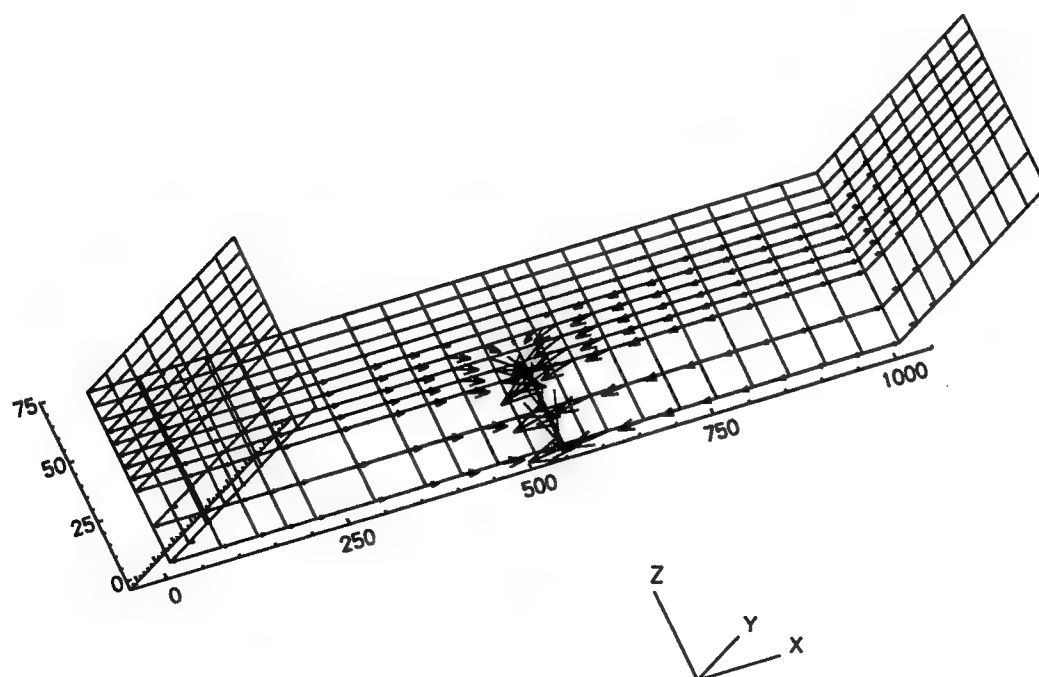


Figure 4.10b Velocity distribution on the x-z cross section through the well for Example 2

4.4 Example 3: 1-D Channel Water Budget and 3-D Subsurface Flow

The subsurface domain is 600 m × 1100 m × 100 m that is discretized by 330 (= 6 × 11 × 5) hexahedral elements (each element has a size of 100 m × 100 m × 20 m) and 504 (= 7 × 12 × 6) subsurface nodes (Figure 4.11). The domain is assumed to contain the soil with a saturated hydraulic conductivity of 1.1×10^{-2} m/s and a porosity of 0.45. The unsaturated characteristic hydraulic properties of the soil in the column are given as

$$\theta = \theta_s - \frac{\theta_s - \theta_r}{h_b - h_a}(h - h_a) \quad (4.3)$$

and

$$K_r = \frac{\theta - \theta_r}{\theta_s - \theta_r} \quad (4.4)$$

where h_b and h_a are the parameters used to compute the water content and the relative hydraulic conductivity, respectively.

To reflect the soil property function given by Eqs. (4.3) and (4.4), we have to modify the subroutine SPROP3 in the source code. For this example, to the following form for computing the moisture content and water capacity.

```

WCR=THPROP(1,MTYP)
WCS=THPROP(2,MTYP)
HAA=THPROP(3,MTYP)
HAB=THPROP(4,MTYP)
DO 390 KG=1,8
HNP=HKG(KG)
HNP=-HNP
IF(HNP.LE.0) GO TO THEN
C
C ----- SATURATED CONDITION
C
  TH(KG,M)=WCS
  DTH(KG,M)=0.0D0
  USKFCT=1.0D0
C
  ELSE
C
C ----- UNSATURATED CASE
C
  THMKG=WCS-(WCS-WCR)*(-HNP-HAA)/(HAB-HAA)
  TH(KG,M)=THMKG
  DTH(KG,M)=-(WCS-WCR)/(HAB-HAA)
  USKFCT=(THMKG-WCR)/(WCS-WCR)

```

END IF

The initial conditions are assumed as a total head of -2 m imposed on the front surface (i.e., $y = 0$ m) of the domain and decreased by 0.1 m for each 100 m increment in the y-direction. The boundary conditions are given as: no flux is imposed on the left (i.e., $x = 0$ m), bottom (i.e., $z = -100$ m), right (i.e., $x = 600$ m), and back (i.e., $y = 1100$ m) surfaces of the domain; total head is held at -2 m on the front surface; and the Variable boundary condition is used on the top surface (i.e., $z = 0$ m) of the domain with a ponding depth of 0 m, minimum pressure of -10 m, and rainfalls of 1.0×10^{-8} m/s and 2.0×10^{-8} m/s on areas A and B, respectively (Figure 4.11). The pointwise iteration method is used to solve the resulting matrix equation. Thus, we do not need the subregion data. A constant time step size of 3,600 seconds is used with 20 time steps to preform a 72,000-second simulation.

Because channel are considered to be line sources/sinks to the subsurface system, channels dwell on the top surface on the subsurface domain (Figure 4.12). The numbers shown on Figure 4.12 are channel reach node numbers. Table 4.1 defines channel reaches with their upstream and downstream reach nodes. Each reach can include more than one channel elements for coordinating with the discretization of subsurface domain and describing channel geometry more precisely. Figure 4.13 displays all 29 channel element nodes for this problem, while Table 4.2 defines the associated 28 channel elements with their upstream and downstream channel element nodes. In Table 4.2, the reach containing channel elements are also indicated. For example, the 7-th channel element is in the 3-rd channel reach. Table 4.3 lists channel element nodes and their associated subsurface nodes that are on the top surface of the subsurface domain.

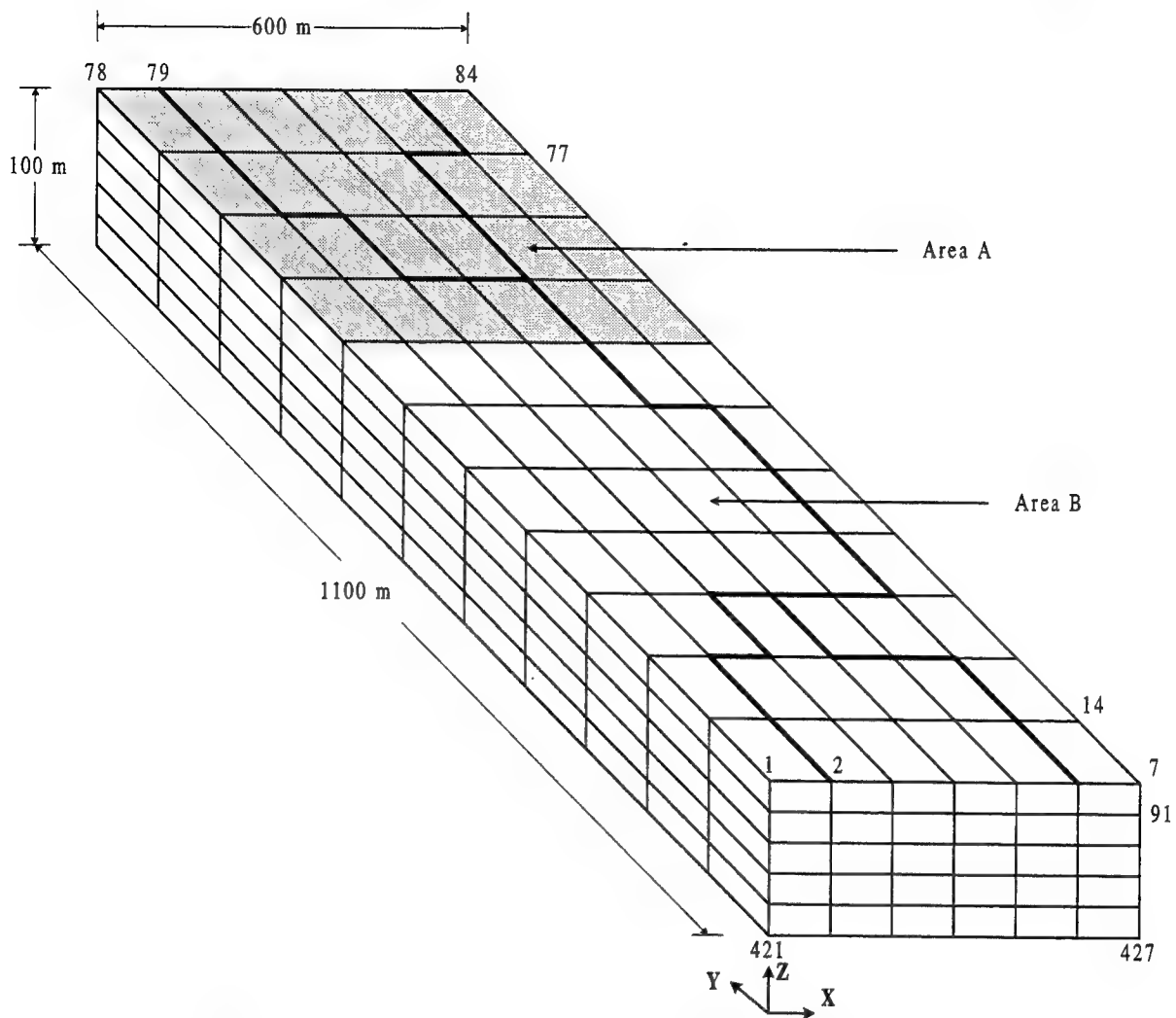
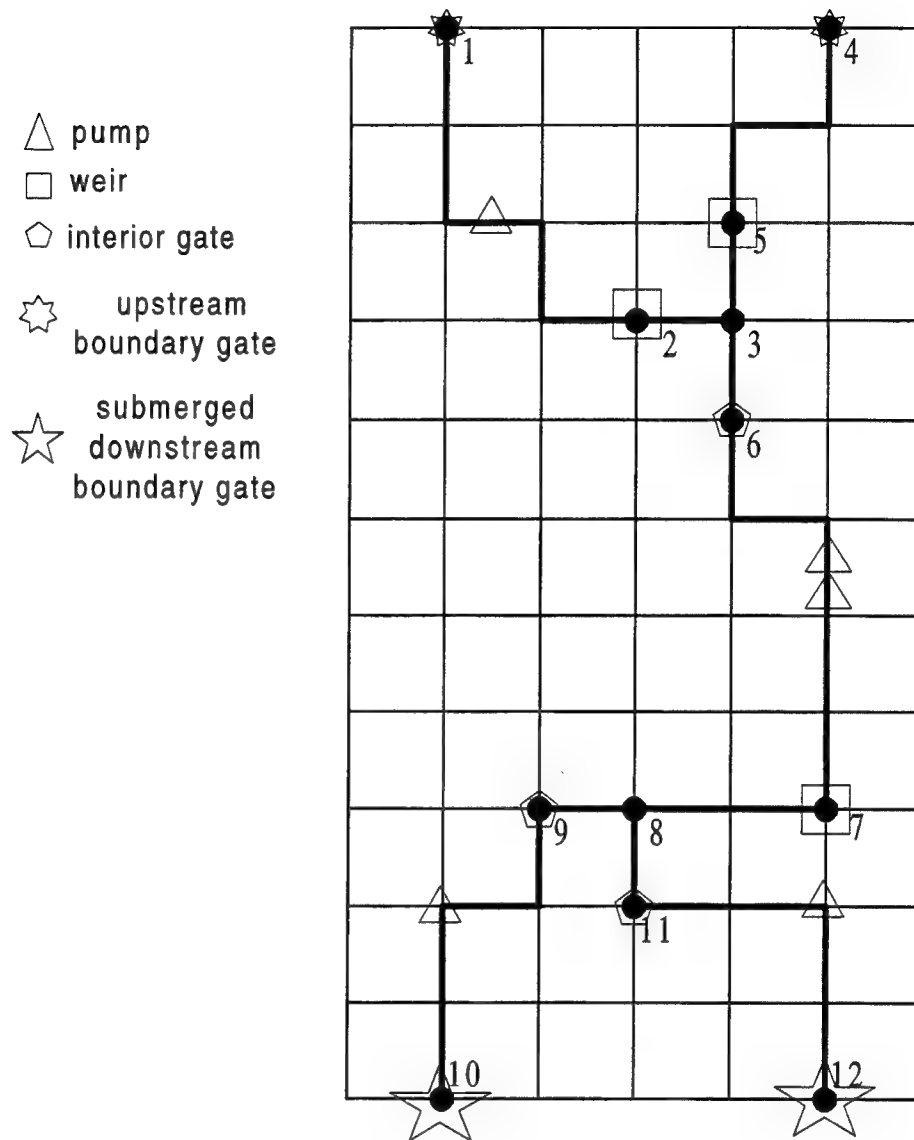


Figure 4.11 The subsurface domain and discretization for Example 3



Channel Reach Nodes

Figure 4.12 The channel reach nodes for Example 3

Table 4.1 Definition of channel reaches for Example 3

Channel Reach No.	Upstream Reach Node	Downstream Reach Node
1	1	2
2	2	3
3	4	5
4	5	3
5	3	6
6	6	7
7	7	8
8	8	9
9	9	10
10	8	11
11	11	12

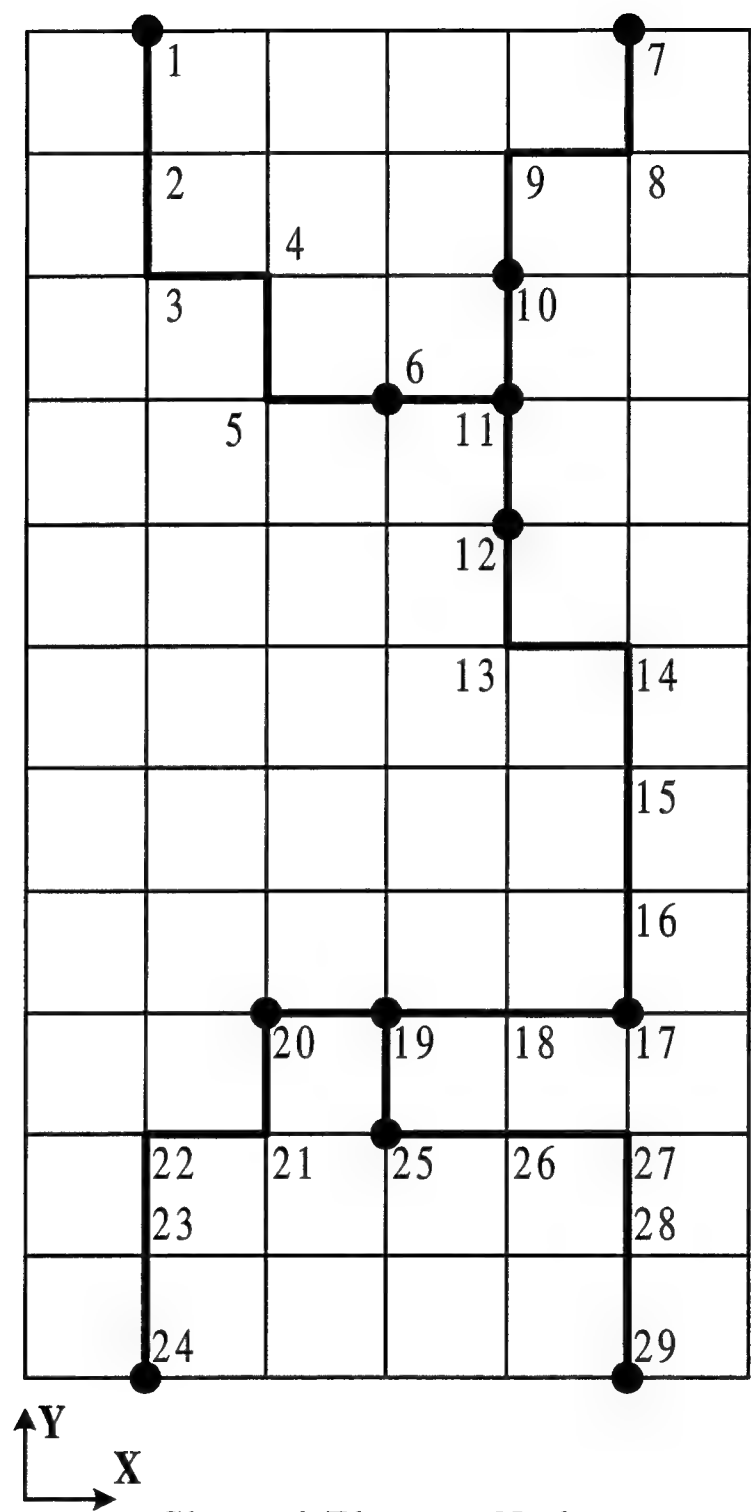


Figure 4.13 Channel element nodes for Example 3

Table 4.2 Definition of channel elements for Example 3

Ch. Ele. No.	Up. Ch. Ele. Node	Down. Ch. Ele. Node	Ch. Reach to Include This Ch. Ele.
1	1	2	1
2	2	3	1
3	3	4	1
4	4	5	1
5	5	6	1
6	6	11	2
7	7	8	3
8	8	9	3
9	9	10	3
10	10	11	4
11	11	12	5
12	12	13	6
13	13	14	6
14	14	15	6
15	15	16	6
16	16	17	6
17	17	18	7
18	18	19	7
19	19	20	8
20	20	21	9
21	21	22	9
22	22	23	9
23	23	24	9
24	19	25	10
25	25	26	11
26	26	27	11
27	27	28	11

28	28	29	11
----	----	----	----

Table 4.3 Channel element nodes and their associated subsurface nodes for Example 3

Channel Element Node	Associated Subsurface Node	Channel Element Node	Associated Subsurface Node
1	79	16	34
2	72	17	27
3	65	18	26
4	66	19	25
5	59	20	24
6	60	21	17
7	83	22	16
8	76	23	9
9	75	24	2
10	68	25	18
11	61	26	19
12	54	27	20
13	47	28	13
14	48	29	6
15	41	-----	-----

Among the 12 channel reach nodes, the 3-th and 8-th nodes are structure-free joints that generate two reach groups: reaches 2, 4, and 5 and reaches 7, 8, and 10, respectively. Since all the reaches in any reach group are freely connected and equilibrium is assumed to be reached instantaneously, the water stages of channel

reaches 2, 4, and 5 are of one value, so are those of channel reaches 7, 8, and 10. The 1-st and 4-th reach nodes are upstream boundary gates with a constant flow rate of $10 \text{ m}^3/\text{s}$ for each. The 10-th and 12-th reach nodes are submerged downstream boundary gates with a tailwater stage of -2 m for both gates. The discharge rates for these two downstream gates and three interior gates (i.e., reach nodes 6, 9, and 11) are described by same analytical functions. Two analytical functions, one for the submerged case and one for the non-submerged case, form a complete set of flow rate profile for interior gates and have been mentioned in section 3.3 (Eqs. (3.9) and (3.10)). As for the two downstream gates, only the submerged analytical function (i.e., Eq. (3.10)) is needed. The flow rates for three weirs (i.e., reach nodes 2, 5, and 7) are calculated by the same formulae as stated in section 3.3 (Eqs. (3.11) and (3.12)).

In this example, the upstream boundary gates at reach nodes 1 and 4 are always open to supply water to the canal and the downstream boundary gates at reach nodes 10 and 12 are closes when the water stage of connected reach is less than 3.0 m .

There are 5 channel pumps installed. They are installed in the 1st, 6-th, 6-th, 9-th, and 11-th reach and each pump is associated with a downstream reach node. The 1-st channel pump is to pump water from/to the 310-th subsurface node, the 2-nd pump from/to the 292-nd subsurface node, the 3-rd pump from/to the 9-th channel reach, the 4-th pump from/to the 264-th subsurface node, and the 5-th pump from/to the 263-th subsurface node.

The upper and lower limits to control the open/close of interior and downstream gates as well as to control the on/off of channel pumps are 3 m and 2 m , respectively, for each reach. For all 28 channel elements, the bottom width is 5 m , the bottom elevation is 0.2 m , both the right and left side slopes are 0.1 , the perimeter thickness is 0.2 m , the perimeter conductivity is 10^{-3} m/s , and the bank elevation is 4 m .

Precipitation for channel reaches is set in accordance with those for the subsurface system. The initial water stage is 2.5 m for all reaches. All gates, except for the two upstream gates, are initially closed and all

channel pumps are initially off. The channel time step size is set to 12 s ($= 3,600 / 300$) which may be adjusted in computation based on the given allowed water depth change in reaches during one channel time step. This allowed water depth change is 0.05 m here.

In the 3-D subsurface simulation, the pressure head tolerance is 2×10^{-2} m for nonlinear iteration and is 10^{-2} m for pointwise iteration. The relaxation factors for both the nonlinear iteration and pointwise iteration are 1.0. In the coupling iteration, 2×10^{-2} m and 2×10^{-2} m are used to determine convergence for the subsurface and the channel, respectively.

To execute the problem, the maximum control-integers in the main program should be specified as follows:

```

C
C ----- 1. FOR BOTH 3-D FLOW AND 3-D TRANSPORT
C
  PARAMETER(MXSNP=504,MXSEL=330,MXSBNP=304,MXSBES=302)
  PARAMETER(MXSTUBS=4*MXSBNP)
  PARAMETER(MXSJBD=27,MXSKBD=8,MXSNTI=20,MXSDTC=1)
  PARAMETER(LTMXSNP=1,LMXSNP=1,LMXSBNP=1,MXSREGN=1)
  PARAMETER(MXSMAT=1,MXSPPM=4,MXSMPPM=14,MXSRMP=8)
C
C ----- 2. FOR FLOW SOURCE/SINK AND BOUNDARY CONDITIONS
C
  PARAMETER(MXSSELF=1,MXSPPRF=1,MXSDDPF=1,MXSWNPF=1,MXSWPRF=1,
>    MXSWDPF=1)
  PARAMETER(MXSCNPF=1,MXSCESF=1,MXSCPRF=1,MXSCDPF=1)
  PARAMETER(MXSNNPF=1,MXSNESEF=1,MXSNPRF=1,MXSNDPF=1)
  PARAMETER(MXSVPNF=84,MXSVESEF=66,MXSVPRF=2,MXSVDPF=2)
  PARAMETER(MXSDNPF=42,MXSDPRF=1,MXSDDPF=2)
  PARAMETER(MXSRNPF=1,MXSRPRF=1,MXSRDPF=1,MXSRMAF=1,MXSRESF=1)
C
C ----- 3. FOR TRANSPORT SOURCE/SINK AND BOUNDARY CONDITIONS
C
  PARAMETER(MXSSELT=1,MXSPPRT=1,MXSDDPT=1,MXSWNPT=1,MXSWPRT=1,
>    MXSWDPT=1)
  PARAMETER(MXSCNPT=1,MXSCEST=1,MXSCPRT=1,MXSCDPT=1)
  PARAMETER(MXSNNPT=1,MXSNEST=1,MXSNPRT=1,MXSNDPT=1)
  PARAMETER(MXSVPNT=1,MXSVEST=1,MXSVPRT=1,MXSVDPPT=1)
  PARAMETER(MXSDNPT=1,MXSDPRT=1,MXSDDPT=1)
C
  PARAMETER(MXSNPW=1,MXSELW=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR 1-D CANAL FLOW
C *****
C

```

```

PARAMETER(MXCNRN=12,MXCNR=11)
PARAMETER(MXCEL=28,MXCNP=29)
PARAMETER(MXCCPM=5)
PARAMETER(MXCGPR=3,MXCGPD=9)
PARAMETER(MXCWPR=3,MXCWPD=9)
PARAMETER(MXCRPR=2,MXCRPD=2)
PARAMETER(MXCPPR=1,MXCPPD=2)
PARAMETER(MXCUPR=2,MXCUPD=2)
PARAMETER(MXCDPR=2,MXCDPD=2)
PARAMETER(MXCNTC=10000)
PARAMETER(MXCNR=2,MXCANG=1)
PARAMETER(MXCHMXR=1,MXCHMXD=2)
PARAMETER(MXCHMNR=1,MXCHMND=2)
PARAMETER(MXCSNP=29,MXCSEL=28)
PARAMETER(MXCNGO=2)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR RETENTION POND
C *****
C
PARAMETER(MXPONK=1,MXPPRK=1,MXDPPK=1,MXPOBK=1,MXOULK=1)
PARAMETER(MXFXPK=1,MXFXDK=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR 2-D OVERLAND FLOW
C *****
C
PARAMETER(MXOVTK=1,MXONTI=1,MXODTC=1)
PARAMETER(MXONP=1,MXOEL=1,MXOBNP=1,MXOBES=1)
PARAMETER(MXOHBP=1,MXOJBD=1,MXOKBD=1)
PARAMETER(MXORES=MXOEL,MXORPR=1,MXORDP=1)
PARAMETER(MXODNP=1,MXODPR=1,MXODDP=1)
PARAMETER(MXOCNP=1,MXOCES=1,MXOCPR=1,MXOCDP=1)
PARAMETER(MXONNP=1,MXONES=1,MXONPR=1,MXONDP=1)
PARAMETER(MXOMAT=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR GRAPHIC INTERFACE
C *****
C
PARAMETER(MXXYPK=9,MXXYSK=9)
PARAMETER(MXPNTK=10,MXLINK=5,MXITPK=30)
PARAMETER(MXPSTK=100)

```

With the above descriptions, the input data can be prepared according to the input guide given Appendix

A. To save space, the input data files and output results are included in the attached floppy disk. Figure 4.14 shows the simulated water stage of reach 1, 6, and 8 for 1-D channel water budget. Figure 4.15 is the simulated

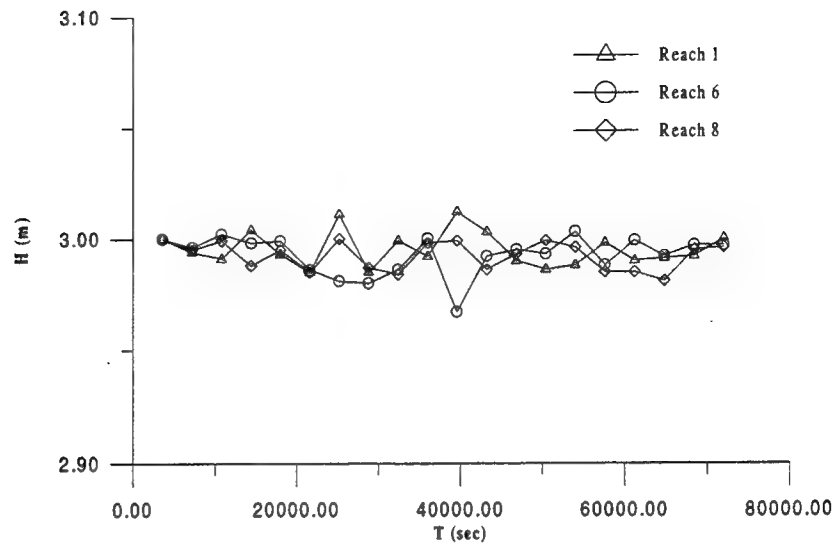


Figure 4.14 Simulated water stage of Reach 1, 6, and 8 for Example 3

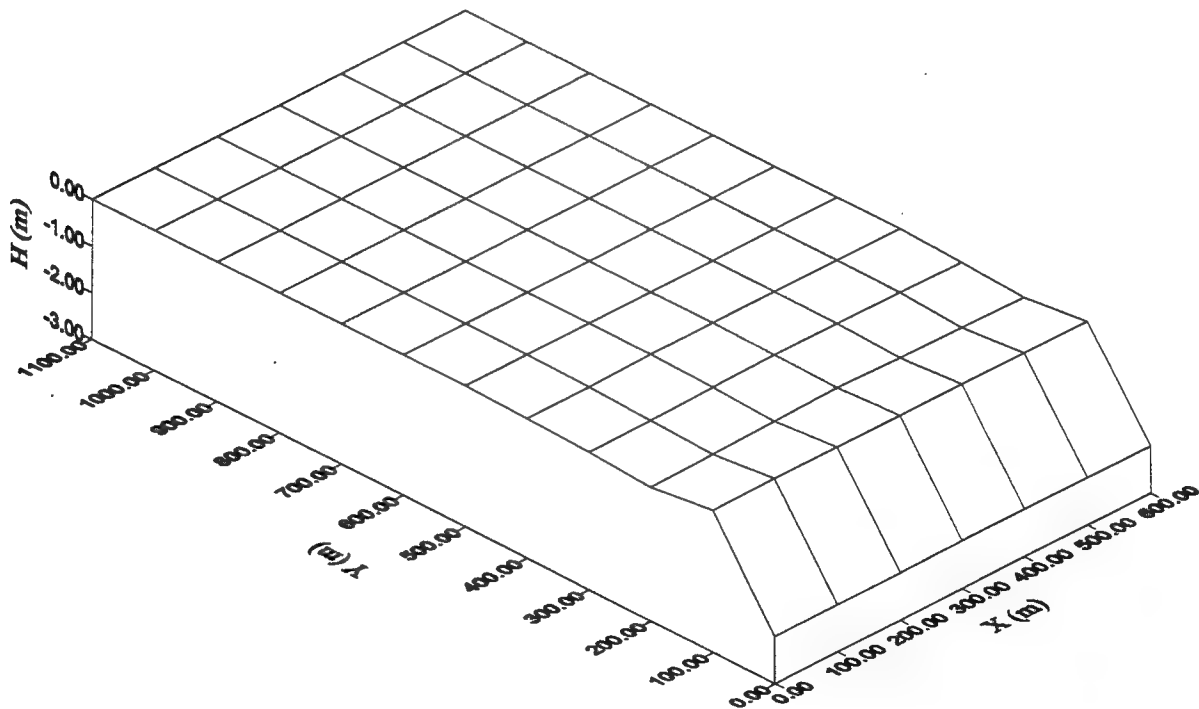


Figure 4.15 Simulated total head on the top surface of 3-D domain for Example 3

total head of the top surface (i.e., $z = 0$ m) for 3-D subsurface flow.

In this example, 1-D canal is operated by the control structures designed to maintain the water stage in a range ($2 \text{ m} < H < 3 \text{ m}$), while interacting with the subsurface flow through infiltration. Because the upstream boundary gates are always open, the inflow flux rate is a constant value of $20 \text{ m}^3/\text{sec}$ ($= 2 \times 10$). For simplifying the estimation, the cross sections of all canal reaches are assumed rectangular with a bottom width of 5 m (given). The total length of canal is 280 (28×10) m and the given initial water stage is 2.5 m. The maximum volume of water that canal needs to reach 3 m of water stage is $700 (280 \times 5 \times 0.5) \text{ m}^3$. During the first subsurface time step (i.e. $t = 3,600 \text{ sec}$), $72,000 (= 20 \times 3,600) \text{ m}^3$ of water flow into the canal. This amount of water is much larger than that the canal can take. As designed, the downstream boundary gates are opened only when the water stage is greater than 3 m. The water exceeding the canal capacity is therefore pumped to the subsurface and the simulated water stage of the canal is slightly varied with 3 m as shown in Figure 4.14 for reaches 1, 6, and 8.

From the output file of this example, the diagnostic results of each reach at the first subsurface time step ($t = 3,600 \text{ sec}$) is listed in the following table.

Reach No.	QTT(J)	QST(J)	QPT(J)	QUT(J)	QDT(J)
1	.288D-4	-.130D00	-.109D+2	.100D+2	.141D+1
2	.576D-5	-.774D-1	.000D00	-.141D+1	.000D00
3	.173D-4	-.796D-1	.000D00	.100D+2	-.967D+1
4	.576D-5	-.774D-1	.000D00	.967D+1	.000D00
5	.576D-5	-.774D-1	.000D00	.000D00	-.795D+1
6	.576D-4	-.122D00	-.588D+1	.794D+1	-.156D+1
7	.231D-4	-.955D-1	.000D00	.156D+1	.000D00
8	.115D-4	-.955D-1	.000D00	.000D00	-.547D00
9	.461D-4	-.953D-1	.235D0+1	.547D00	-.250D+1
10	.115D-4	-.955D-1	.000D00	.000D00	-.616D00

11	.461D-4	-.945D-1	-.412D-1	.616D00	-.175D00
----	---------	----------	----------	---------	----------

QTT(J) is the net volumetric flow rate due to precipitation for the J-th reach. QST(J) is the net volumetric flow rate through the canal-subsurface interface for the J-th reach. QPT(J) is the net volumetric flow rate due to pumping for the J-th reach. QUT(J) is the net volumetric flow rate from its upstream reach for the J-th reach. QDT(J) is the net volumetric flow rate from its downstream reach for the J-th reach. The unit is m^3/sec and "+" means into this reach and "-" means out of this reach.

According to the above table, the precipitation is insignificant when compared with other items. During the first time step, the total amount of water pumped to subsurface is $52,096 (= 3,600 \times \sum QPT(J)) \text{ m}^3$. From the initial condition of the subsurface, we know that there is a 2 m thickness of unsaturated zone with moisture content ranging from 0.44 to 0.45 corresponding to the initial pressure head $h = -2 \text{ m}$ to 0 m . Therefore, the maximum volume of water needed to saturate this zone is $13,200 (= 1,100 \times 600 \times 2 \times 0.01) \text{ m}^3$. As estimated above, the canal has a water stage around 3.0 m and pumps $52,096 \text{ m}^3$ of water to subsurface at the first time step. In the mean time, the subsurface has been saturated because $52,096 \text{ m}^3$ of water pumped to the subsurface is much more than the required $13,200 \text{ m}^3$ for saturation. For each of the rest simulation time steps, the canal may also pump this amount of water to subsurface and the resulting water stage is varied around 3.0 m, because the upstream boundary gate is always open and the downstream boundary gate is only open as the water stage greater than 3 m. Let us assume the pumping volume is $52,096 \text{ m}^3$ for each of 20 subsurface time steps. The difference between the total pumpage $1,041,092 (= 52,096 \times 20) \text{ m}^3$ and $13,200 \text{ m}^3$ is $1,028,720 \text{ m}^3$, which is estimated to flow out of subsurface. From the simulation of COSFLOW, the diagnostic output shows the total volume of water flowing out of subsurface is $1,009,700 \text{ m}^3$. As the results from both the estimation and model simulation of this example, it has demonstrated that the pumping strategy works well to efficiently discharge the extra amount of water in the canal system. Figure 4.15 shows the simulated total head distribution on the ground surface of 3-D domain at $t = 72,000 \text{ sec}$. With this large amount of pumping, most of the subsurface region is

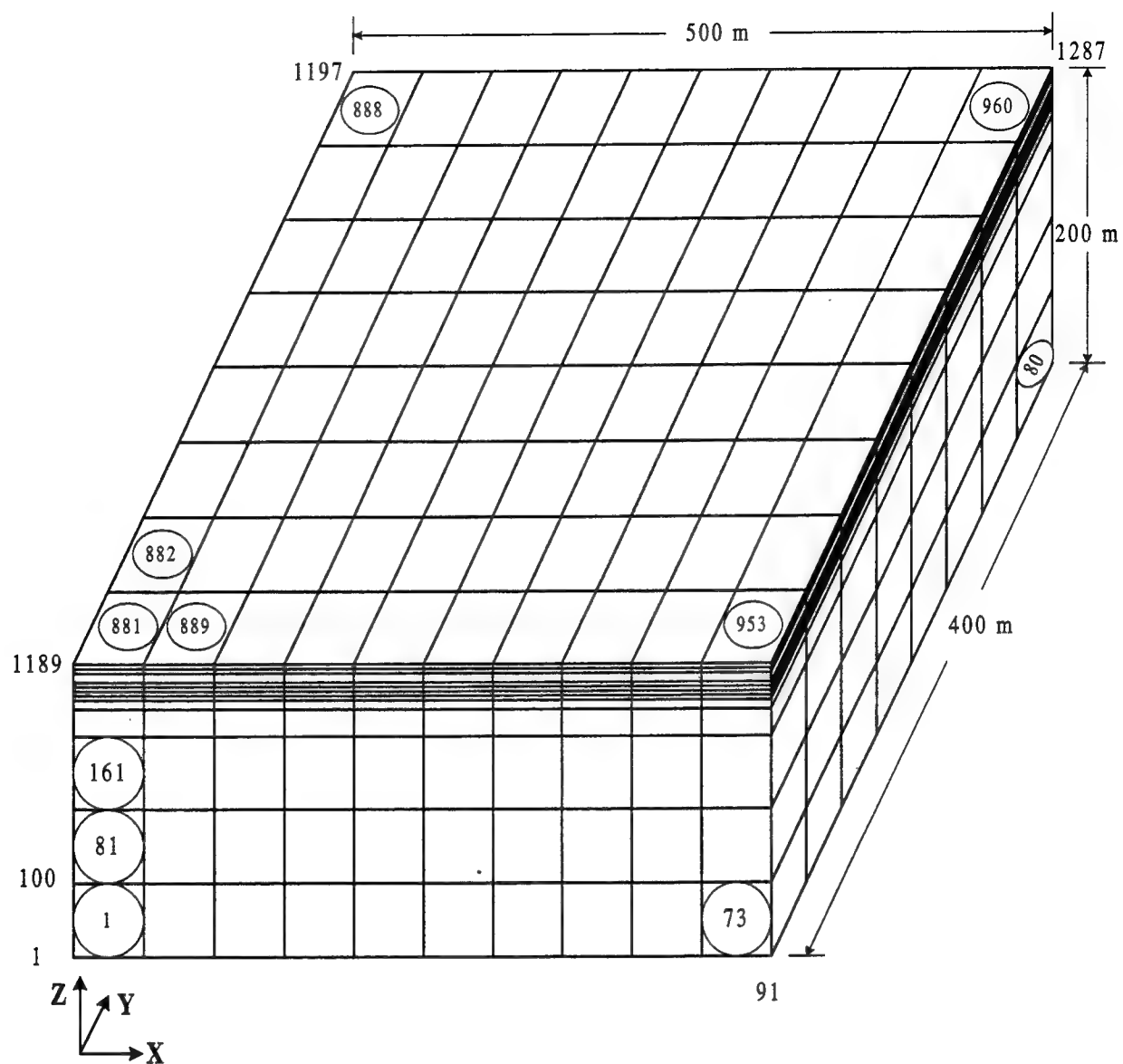
saturated except on the front surface prescribed by a Dirichlet total head -2 m.

4.5 Example 4: 2-D Overland Flow and 3-D Subsurface Flow

The subsurface domain is $500 \text{ m} \times 400 \text{ m} \times 200 \text{ m}$ that is discretized by 960 ($= 10 \times 8 \times 12$) hexahedral elements and 1287 ($= 11 \times 9 \times 13$) nodes (Figure 4.16). The nodes are located at $z = 0, 50, 100, 150, 175, 190, 193, 194, 195, 196, 198, 199$, and 200 m in the z -direction. In the x and y directions, nodes are spaced evenly at $\Delta x = 50 \text{ m}$ and $\Delta y = 50 \text{ m}$. A saturated hydraulic conductivity of $5.5 \times 10^{-4} \text{ m/s}$ and a porosity of 0.45 are used in this example problem. The unsaturated characteristic hydraulic properties of the soil are given as the same as Example 3. The initial conditions of subsurface assume a total head of 195 m through the whole region. The boundary conditions of subsurface are given as: no flux is imposed on the front (i.e., $y = 0 \text{ m}$), back (i.e., $y = 400 \text{ m}$), left (i.e., $x = 0 \text{ m}$), and right (i.e., $x = 500 \text{ m}$) surface of the domain; and total head is held at 195 m on the bottom surface.

On the top of subsurface is fully covered by the domain of 2-D overland which contains 80 quadrilateral elements ($= 10 \times 8$) and 99 nodes ($= 11 \times 9$) (Figure 4.17). The initial condition of overland is assumed a water depth of 0.0 m . The boundary conditions are given as: no flux is imposed on the front (i.e., $y = 0 \text{ m}$), back (i.e., $y = 400 \text{ m}$), left (i.e., $x = 0 \text{ m}$), and right (i.e., $x = 500 \text{ m}$) sides of the domain; and a rainfall of $4.0 \times 10^{-5} \text{ m/sec}$ on the upper half plane of the domain (i.e. $y = 200 \text{ m}$ to 400 m) from $t = 0 \text{ sec}$ to $9,000 \text{ sec}$.

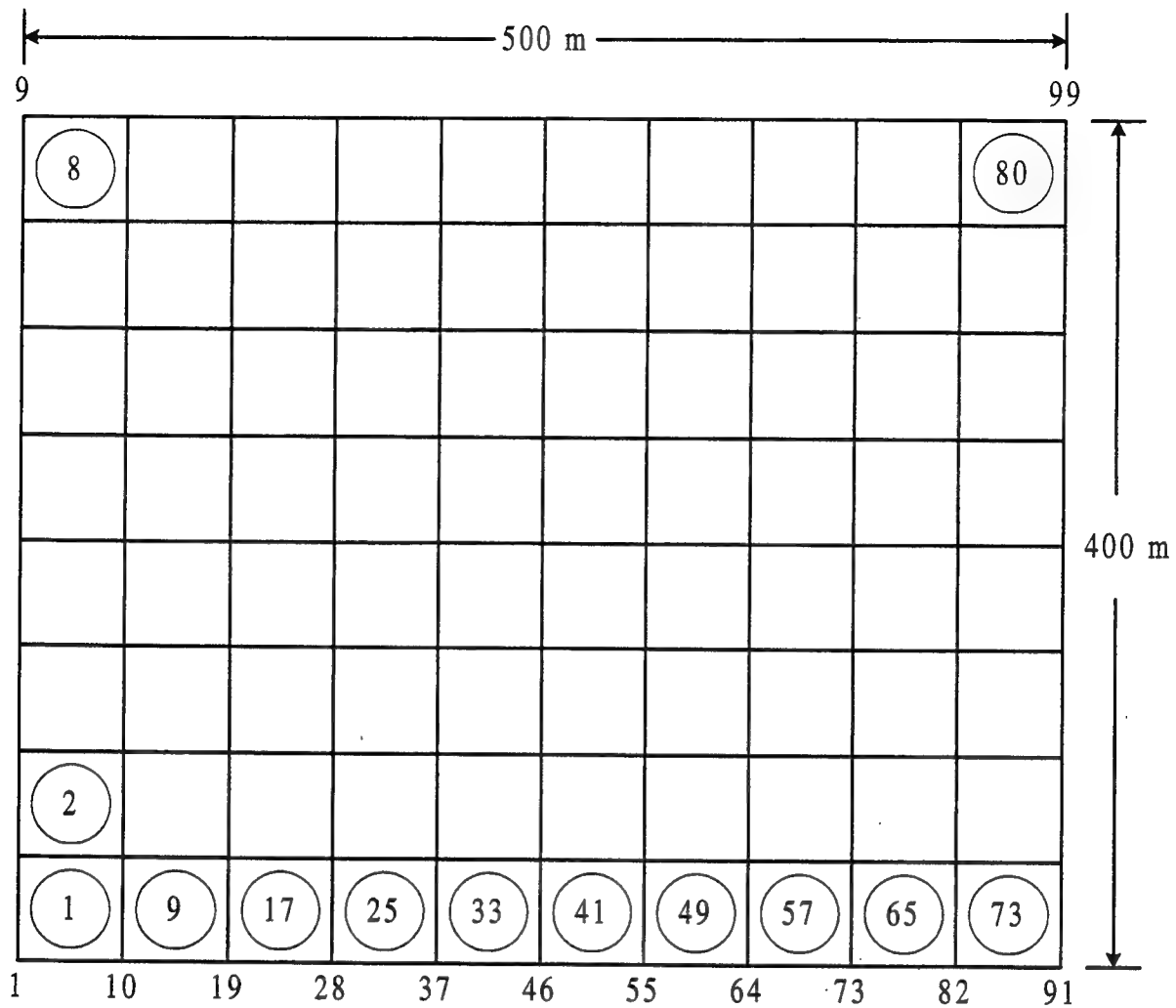
A total of 20 time steps ($27,000 \text{ sec}$) for subsurface is simulated and each of them is divided into 9 time steps for overland flow simulation. From the subsurface point of view, the time-step size of $1,800 \text{ sec}$ is used for the first 5 time steps, then it is changed to 900 sec for the next 10 time steps and $1,800 \text{ sec}$ for the last 5 time steps. Therefore, the time-step size for overland is 200 sec (45 overland time steps), then 100 sec (90 overland



1189 = the 1189-th Subsurface node

81 = the 81-th Subsurface element

Figure 4.16 The subsurface domain and its discretization for Example 4



37 = the 37-th Overland node

(25) = the 25-th Overland element

Figure 4.17 The 2-D overland domain and its discretization for Example 4

time steps), and 200 sec (45 overland time steps).

Both of the pressure head tolerance for 3-D and water depth tolerance for 2-D are 2×10^{-3} m for nonlinear iteration. The relaxation factors for nonlinear iteration are 0.5 and 1.0 for 3-D and 2-D simulations, respectively.

To execute the problem, the maximum control-integers in the including file should be specified as:

```

C
C ----- 1. FOR BOTH 3-D FLOW AND 3-D TRANSPORT
C
  PARAMETER(MXSNP=1287,MXSEL=960,MXSBNP=594,MXSBES=592)
  PARAMETER(MXSTUBS=4*MXSBNP)
  PARAMETER(MXSJBD=27,MXSKBD=8,MXSNTI=20,MXSDTC=4)
  PARAMETER(LTMXSNP=1,LMXSNP=1,LMXSBW=1,MXSREGN=1)
  PARAMETER(MXSMAT=1,MXSSPPM=4,MXSMPPM=14,MXSRMP=8)
C
C ----- 2. FOR FLOW SOURCE/SINK AND BOUNDARY CONDITIONS
C
  PARAMETER(MXSSSELF=1,MXSSPRF=1,MXSSDPF=1,MXSWNPF=1,MXSWPRF=1,
>    MXSWDPF=1)
  PARAMETER(MXSCNPF=1,MXSCESF=1,MXSCPRF=1,MXSCDPF=1)
  PARAMETER(MXSNNPF=1,MXSNEF=1,MXSNPRF=1,MXSNDPF=1)
  PARAMETER(MXSVNPF=1,MXSVESF=1,MXSVPRF=1,MXSVDPF=1)
  PARAMETER(MXSDNPF=100,MXSDPRF=1,MXSDDPF=2)
  PARAMETER(MXSRNPF=1,MXSRPRF=1,MXSRDPF=1,MXSRMAF=1,MXSRESF=1)
C
C ----- 3. FOR TRANSPORT SOURCE/SINK AND BOUNDARY CONDITIONS
C
  PARAMETER(MXSELST=1,MXSSPRT=1,MXSSDPT=1,MXSWNPT=1,MXSWPRT=1,
>    MXSWDPT=1)
  PARAMETER(MXSCNPT=1,MXSCEST=1,MXSCPRT=1,MXSCDPT=1)
  PARAMETER(MXSNNPT=1,MXSNEST=1,MXSNPRT=1,MXSNPT=1)
  PARAMETER(MXSVNPT=1,MXSVEST=1,MXSVPRT=1,MXSVPT=1)
  PARAMETER(MXSDNPT=1,MXSDPRT=1,MXSDPT=1)
C
  PARAMETER(MXSNPW=1,MXSELW=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR 1-D CANAL FLOW
C *****
C
  PARAMETER(MXCNRN=1,MXCNR=1)
  PARAMETER(MXCEL=1,MXCNP=1)
  PARAMETER(MXCCPM=1)
  PARAMETER(MXCGPR=1,MXCGPD=1)
  PARAMETER(MXCWPR=1,MXCWPD=1)
  PARAMETER(MXCRPR=1,MXCRPD=1)
  PARAMETER(MXCPPR=1,MXCPPD=1)
  PARAMETER(MXCUPR=1,MXCUPD=1)
  PARAMETER(MXCDPR=1,MXCDPD=1)
  PARAMETER(MXCNTC=1)
  PARAMETER(MXCNJR=1,MXCANG=1)

```

```

PARAMETER(MXCHMXR=1,MXCHMXD=1)
PARAMETER(MXCHMNR=1,MXCHMND=1)
PARAMETER(MXCSNP=1,MXCSEL=1)
PARAMETER(MXCNGO=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR RETENTION POND
C *****
C
PARAMETER(MXPONK=1,MXPPRK=1,MXDPPK=1,MXPOBK=1,MXOULK=1)
PARAMETER(MXFXPK=1,MXFXDK=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR 2-D OVERLAND FLOW
C *****
C
PARAMETER(MXOVTK=9,MXONTI=200,MXODTC=4)
PARAMETER(MXONP=99,MXOEL=80,MXOBNP=36,MXOBES=36)
PARAMETER(MXOHBP=12,MXOJBD=9,MXOKBD=6)
PARAMETER(MXORES=MXOEL,MXORPR=2,MXORDP=4)
PARAMETER(MXODNP=1,MXODPR=1,MXODDP=1)
PARAMETER(MXOCNP=1,MXOCES=1,MXOCPR=1,MXOCDP=1)
PARAMETER(MXONNP=1,MXONES=1,MXONPR=1,MXONDP=1)
PARAMETER(MXOMAT=1)
C
C *****
C ===== MAXIMUM DIMENSION BLOCK FOR GRAPHIC INTERFACE
C *****
C
PARAMETER(MXXYPK=4,MXXYSK=3)
PARAMETER(MXPNTK=10,MXLINK=5,MXITPK=30)
PARAMETER(MXPSTK=100)

```

With the above descriptions, the input data can be prepared according to the instructions in Appendix

A. To save space, the input data files and output results are given in the attached floppy disk. Figure 4.18 and Figure 4.19 depict the simulated water depth of overland flow at $t = 1,800$ sec and $7,200$ sec as a result of a non-uniform rainfall and at $t = 16,200$ sec after rainfall stopped at $t = 9,000$ sec. Figure 4.20 is the pressure head distribution of on the y-z subsurface cross section across $x = 250$ m at $t = 9,000$ sec, $9,900$ sec, and $18,000$ sec.

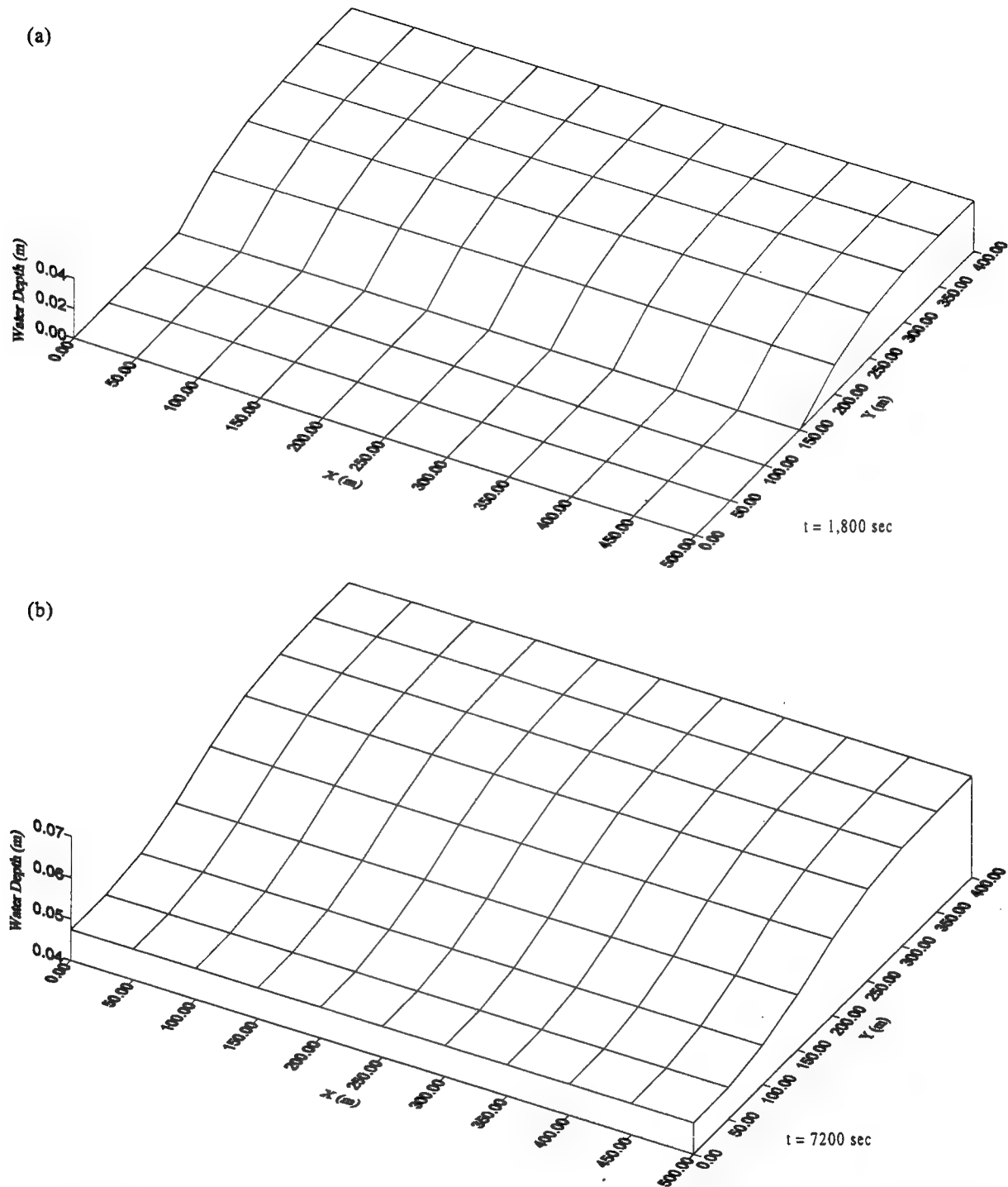


Figure 4.18 Simulated overland water depth at (a) $t = 1,800 \text{ sec}$ and (b) $t = 7,200 \text{ sec}$ for Example 4

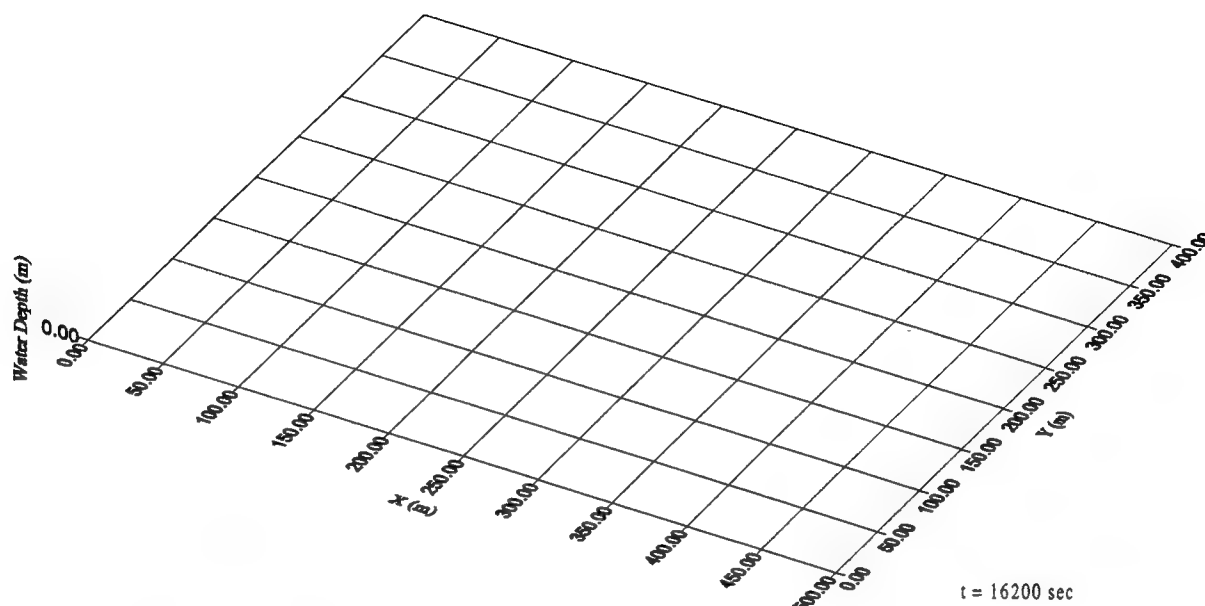


Figure 4.19 Simulated overland water depth at $t = 16,200$ sec for Example 4

The example is designed as: (1) the rainfall continuously falls on the upper half region of 2-D overland domain from $t = 0$ to 9,000 sec, (2) the bottom elevation of 2-D overland domain is uniform, (3) there is no sinks except infiltration to the subsurface, and (4) the boundary condition of subsurface domain is impermeable except a constant Dirichlet head (195 m) is imposed on the bottom surface. Therefore, the distribution of water depth over the 2-D overland domain is controlled by the non-uniform rainfall and the infiltration into the subsurface. The infiltration rate mentioned here is calculated based on the information of water depth on the overland region, the rainfall rate, the soil property of the porous media, and the pressure head distribution in the subsurface. The example is designed as: (1) the rainfall continuously falls on the upper half region of 2-D overland domain from $t = 0$ to 9,000 sec, (2) the bottom elevation of 2-D overland domain is uniform, (3) there is no sinks except infiltration to the subsurface, and (4) the boundary condition of subsurface domain is impermeable except a constant Dirichlet head (195 m) is imposed on the bottom surface. Therefore, the distribution of water depth over

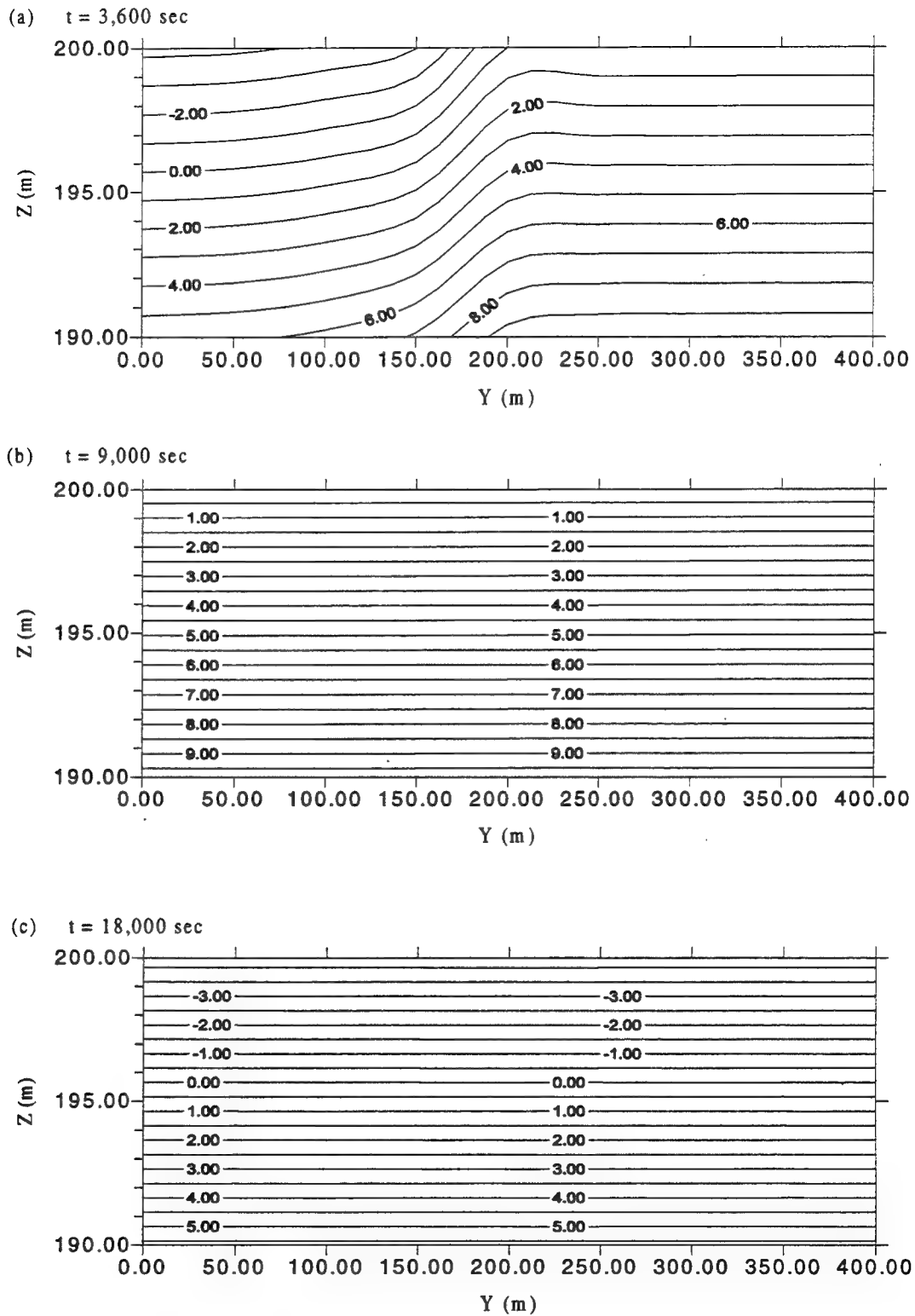


Figure 4.20 Simulated subsurface pressure head on the y-z plane across $x = 250 \text{ m}$ at (a) $t = 3,600 \text{ sec}$, (b) $t = 9,000 \text{ sec}$, and (c) $t = 18,000 \text{ sec}$

the 2-D overland domain is controlled by the non-uniform rainfall and the infiltration into the subsurface. The infiltration rate mentioned here is calculated based on the information of water depth on the overland region, the rainfall rate, the soil property of the porous media, and the pressure head distribution in the subsurface. Also, the computation strategy of one subsurface time-step lag is employed in calculating the infiltration rate between overland and subsurface regions. For example, the infiltration rate needed for the n -th subsurface time step is calculated based on the pressure head distribution of subsurface at the n -th subsurface time step and the water depth distribution on the overland at the $(n-1)$ -th subsurface time step. A more detailed description about the implementation can be found in Appendix B.2.3.

Figure 4.18(a) shows the water depth distribution on 2-D overland domain at $t = 1,800$ sec. The water distributes only on the upper half part of the domain but the lower half part is dry. This is because the rainfall falls on $y = 200$ m to 400 m only. Once the water is accumulated on the upper region, the water will flow toward the lower region due to the water stage gradient.

At $t = 7,200$ sec, the total volume of water falling on the 2-D overland is $28,800 \text{ m}^3 (= 4 \times 10^{-5} \times 500 \times 200 \times 7,200)$. In the subsurface domain, there is a 5 m thickness of unsaturated zone with the moisture content ranging from 0.437 to 0.45 corresponding to the initial pressure head $h = -5$ m to 0 m. The maximum volume of water needed to fill the void volume of unsaturated zone is $13,000 (= 5 \times 500 \times 400 \times 0.013) \text{ m}^3$, which is smaller than the amount of water from rainfall. Therefore, water accumulated on the overland regime is still significant. At $t = 7,200$ sec (Figure 4.18(b)), the upper half part of 2-D overland region has a higher water depth than the lower half part because the rainfall falls on $y = 200$ m to 400 m only.

After rainfall stops at $t = 9,000$ sec, the water gradually infiltrates into the subsurface because there is a constant Dirichlet head of 195 m imposed on the bottom surface of 3-D domain. Figure 4.19 depicts there is no water on the 2-D domain at $t = 16,200$ sec since the water on the 2-D domain infiltrates into the subsurface and then flows out of subsurface in order to maintain the imposed boundary condition on the bottom surface.

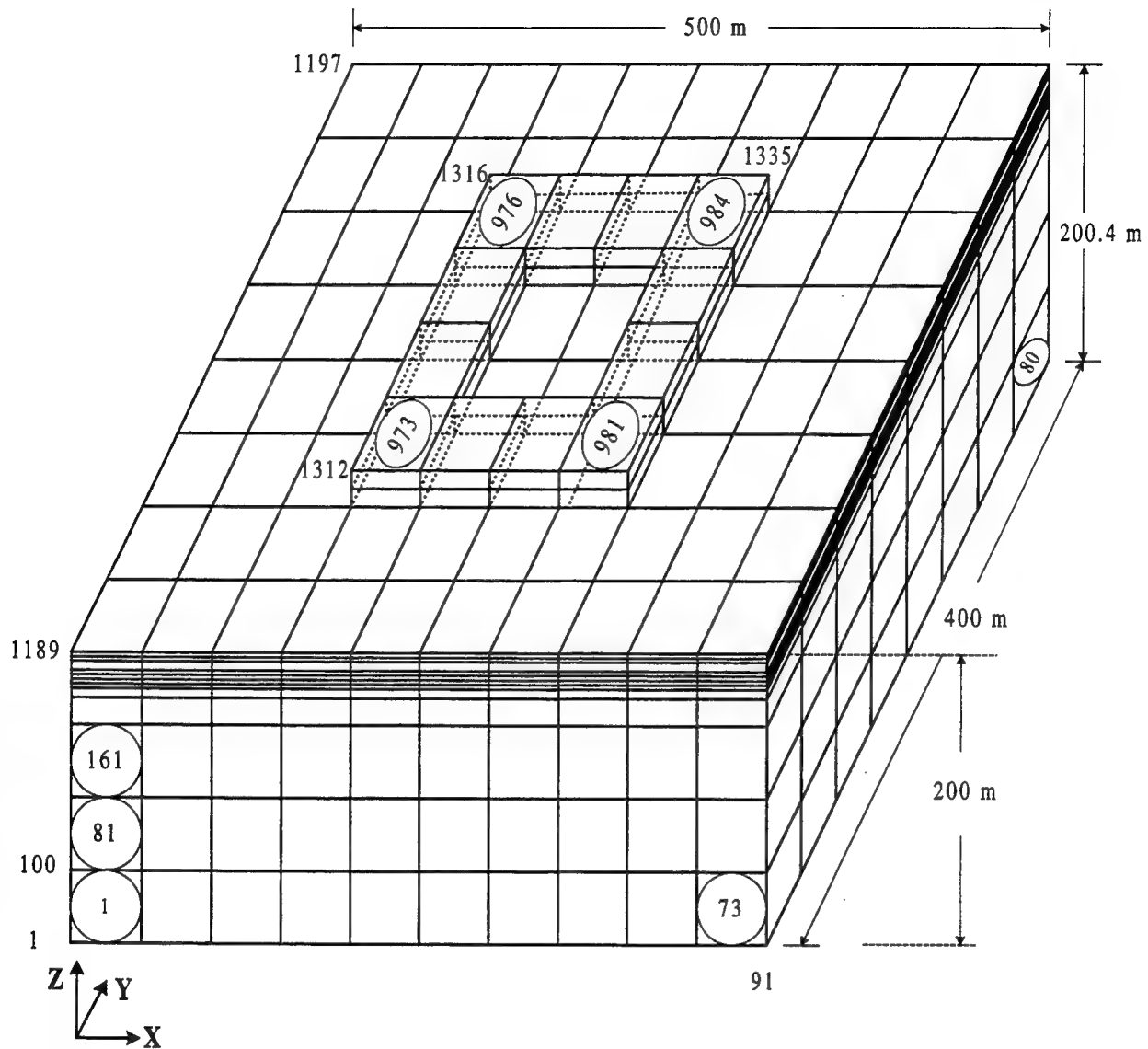
Figure 4.20 is the simulated pressure head distribution on the y-z cross section at $x = 250$ m for three different times $t = 3,600$ sec, $9,000$ sec, and $18,000$ sec. Figure 4.20(a) shows that only half of the region is saturated at $t = 3,600$ sec because the rainfall only falls on the upper half domain ($y = 200$ m to $y = 400$ m). Due to the one time-step lag in the calculation of the infiltration, the water depth distribution of 2-D domain at $t = 1,800$ sec (Figure 4.18(a)) completely matches the pressure head distribution of 3-D domain at $t = 3,600$ sec (Figure 4.20(a)).

Figure 4.20(b) depicts that the whole subsurface region is saturated and has a uniformly distributed pressure head at $t = 9,000$ sec because the accumulated amount of water from rainfall is large enough to saturate the 3-D domain. As mentioned above, the 2-D domain is full of water at $t = 7,200$ sec (Figure 4.18(b)). The result indicates that a saturated pressure head distribution of 3-D domain at $t = 9,000$ sec is expected and shown in Figure 4.20(b).

At $t = 18,000$ sec (Figure 4.20(c)), there is an unsaturated zone right below the ground surface because the rainfall has stopped at $t = 9,000$ sec and the water continuously flows out of the subsurface in order to maintain the prescribed Dirichlet total head of 195 m imposed on the bottom. In Figure 4.19, no water is left on the 2-D domain at $t = 16,200$ sec. The fact reflects the existence of an unsaturated zone in the subsurface domain as shown in Figure 4.20(c).

4.6 Example 5: 1-D Channel Water Budget, 2-D Overland Flow, and 3-D Subsurface Flow

The subsurface domain of this example is almost the same as Example 4, except there is a retention pond of $200 \text{ m} \times 200 \text{ m} \times 2 \text{ m}$ on the center of the top surface and the top surface of 3-D domain has a slope of $1/1000$ from $y = 0$ m to $y = 400$ m. Therefore, the thickness of 3-D region is 200 m on the front and 200.4 m on the back. The 3-D domain is discretized by $984 (= 10 \times 8 \times 12 + 2 \times 12)$ hexahedral elements and $1335 (= 11 \times 9 \times 13 + 2 \times 24)$ nodes (Figure 4.21). A saturated hydraulic conductivity of 5.5×10^{-4} m/s and a porosity of 0.45 are used



1189 = the 1189-th Subsurface node

81 = the 81-th Subsurface element

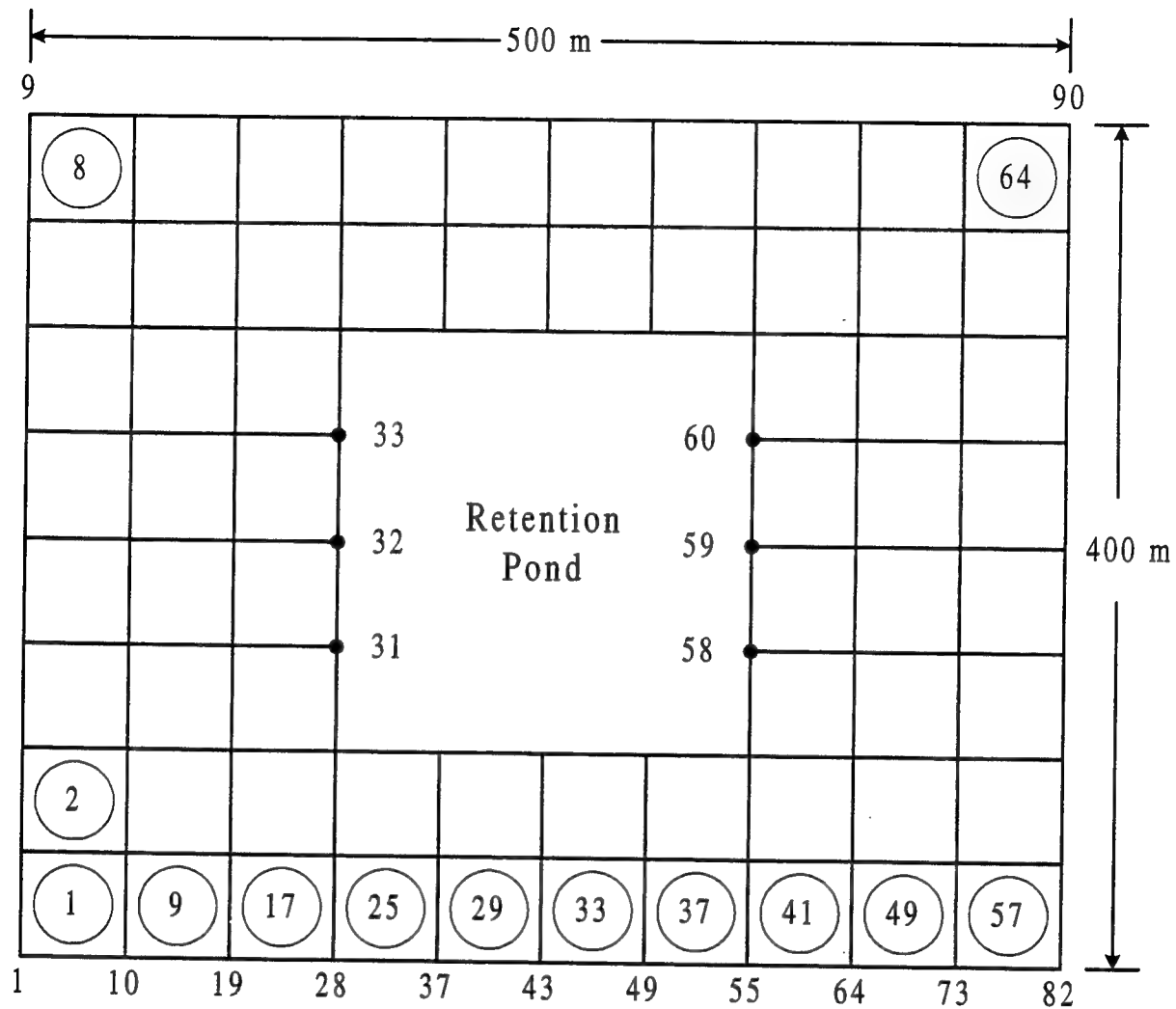
Figure 4.21 The subsurface domain and its discretization for Example 5

in this example problem. The unsaturated characteristic hydraulic properties of the soil are given as the same as Example 3. The initial conditions of subsurface assume a total head of 196 m through the whole region. The boundary conditions of subsurface are given as: no flux is imposed on the front (i.e., $y = 0$ m), back (i.e., $y = 400$ m), left (i.e., $x = 0$ m), and right (i.e., $x = 500$ m) surface of the domain; and total head is held as 196 m on the bottom surface. Because the dike of the retention pond is not included in the 2-D domain, the variable boundary with a rainfall of 2.0×10^{-5} m/sec from $t = 0$ sec to 18,000 sec and 1.0×10^{-5} m/sec from $t = 18,000$ sec to 24,000 sec are imposed on the top surface of the dike (i.e. elements 973 to 984) (Figure 4.21).

The interfaces between the retention pond and the 3-D domain contain 20 elements and 25 nodes from the 3-D point of view. The bottom elevation and the maximum allowed water stage of retention pond are 200 m and 202 m, respectively. The initial condition is assumed as a water stage of 201.5 m. The rainfall rates are also 2.0×10^{-5} m/sec from $t = 0$ sec to 18,000 sec and 1.0×10^{-5} m/sec from $t = 18,000$ sec to 24,000 sec. When the water stage of the retention pond exceeds the maximum allowed value 202 m, the surcharge is assumed to spill to six 2-D nodes evenly (i.e. 31, 32, 33, 58, 59, and 60 as shown in Figure 4.22).

The 2-D overland domain contains 64 quadrilateral elements ($= 10 \times 8 - 4 \times 4$) and 90 nodes ($= 11 \times 9 - 3 \times 3$) (Figure 4.22). The initial condition for the overland region is assumed as a water depth of 0.0 m through the whole domain. The boundary conditions are given as: no flux is imposed on the back (i.e., $y = 400$ m), left (i.e., $x = 0$ m), and right (i.e., $x = 500$ m) sides of the domain; and a rainfall of 2.0×10^{-5} m/sec on the whole region from $t = 0$ sec to 18,000 sec and 1.0×10^{-5} m/sec from $t = 18,000$ sec to 24,000 sec. On the front side (i.e., $y = 0$ m), there is no flux boundary imposed from $x = 0$ m to 150 m and $x = 350$ m to 500 m and a rating curve (same as Example 1B) is used to describe a radiation boundary from $x = 150$ m to 350 m.

The 1-D channel contains 1 reach, 2 reach nodes, 9 channel element nodes, and 8 channel elements (Figure 4.23). The 1-st reach node is a non-submerged upstream boundary gate with constant flow rates of $20 \text{ m}^3/\text{sec}$ from $t = 0$ sec to 18,000 sec and $10^3/\text{sec}$ from $t = 18,000$ sec to 24,000 sec. The 2-nd reach node is a submerged

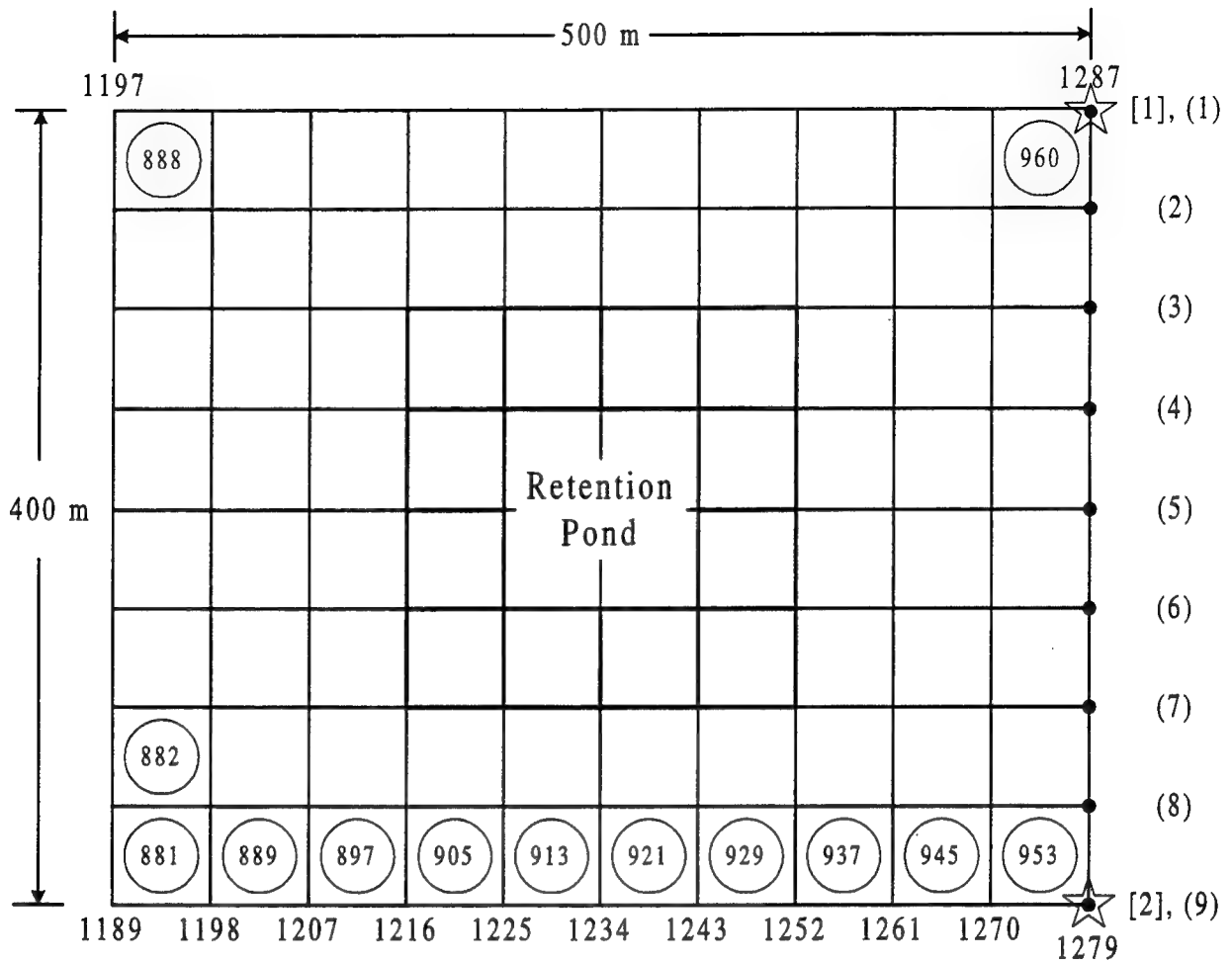


- Overland global node to which water is pumped from the retention pond

37 = the 37-th Overland node

(29) = the 29-th Overland element

Figure 4.22 The overland domain and its discretization for Example 5



☆ Channel reach node , [1] = the 1-st reach node

• Channel element node, (5) = the 5-th element node

1216 = the 1216-th Subsurface node

897 = the 897-th Subsurface element

Figure 4.23 The 1-D canal and its relation with the top of the subsurface domain

downstream boundary gate with a constant tailwater stage of 196 m and its flow rate is calculated by a user-provided formula as mentioned in section 3.3 (Eq. (3.10)).

Only 1 channel pump is installed at the 2-nd reach node for pumping water to the retention pond. The pumping rate is $25 \text{ m}^3/\text{sec}$ and is initially off. The upper and lower limits to control the open/close of downstream gates and the on/off of channel pumps are 203 m and 201 m, respectively. For all 8 channel elements, the bottom width, the bottom elevation, both the right and left side slopes, the perimeter thickness, the perimeter conductivity, and the bank elevation are 4 m, 200.2 m, 0.1, 0.2 m, $1.1 \times 10^{-7} \text{ m/s}$ and 204 m, respectively. Rainfall intensity is $2.0 \times 10^{-5} \text{ m/sec}$ for all channel elements from $t = 0 \text{ sec}$ to 18,000 sec and changes to $1.0 \times 10^{-6} \text{ m/sec}$ for $t = 18,000 \text{ sec}$ to 24,000 sec. The initial water stage is 202 m.

A total of 20 time steps for subsurface is simulated and each of them is divided into 300 time steps for 1-D channel simulation. Every 25 channel time steps is assigned to one time step for overland flow simulation (i.e. 20 time steps for 3-D, 6,000 time steps for 1-D, and 240 time steps for 2-D). From the subsurface point of view, the initial time-step size of 1,800 sec is used for 10 time steps and then it is changed to 600 sec for another 10 time steps. Therefore, the time step size for overland flow simulation is 150 sec for the first 120 time steps and 50 sec for the other 120 time steps. The time-step size for 1-D channel is 6 sec for the first 3,000 time steps and 2 sec for the other 3,000 time steps. During the simulation, the actual 1-D time-step size may be adjusted according to the given allowed water stage change in reaches in a channel time step. This allowed change of water stage is set as 0.05 in this example.

Both of the pressure head tolerance for 3-D and water depth tolerance for 2-D are $2 \times 10^{-3} \text{ m}$ for nonlinear iteration. The relaxation factors for nonlinear iteration are 0.5 and 1.0 for 3-D and 2-D simulations, respectively. In the coupling iteration between 1-D channel and 3-D subsurface, $2 \times 10^{-3} \text{ m}$ is used to determine both of the convergence of water stage for 1-D and pressure head for 3-D.

To execute the problem, the maximum control-integers in the including file should be specified as:

```

C
C ----- 1. FOR BOTH 3-D FLOW AND 3-D TRANSPORT
C
  PARAMETER(MXSNP=1335,MXSEL=984,MXSBNP=900,MXSBES=800)
  PARAMETER(MXSTUBS=4*MXSBNP)
  PARAMETER(MXSJBD=27,MXSKBD=8,MXSNTI=20,MXSDTC=4)
  PARAMETER(LTMXSNP=1,LMXSNP=1,LMXSBW=1,MXSREGN=1)
  PARAMETER(MXSMAT=1,MXSSPPM=4,MXSMPPM=14,MXSRMP=8)
C
C ----- 2. FOR FLOW SOURCE/SINK AND BOUNDARY CONDITIONS
C
  PARAMETER(MXSELF=1,MXSSPRF=1,MXSSDPF=1,MXSWNPF=1,MXSWPRF=1,
>    MXSWDPF=1)
  PARAMETER(MXSCNPF=1,MXSCESF=1,MXSCPRF=1,MXSCDPF=1)
  PARAMETER(MXSNNPF=1,MXSNEF=1,MXSNPRF=1,MXSNDPF=1)
  PARAMETER(MXSVPNF=24,MXSVEF=12,MXSVPRF=1,MXSVDPF=4)
  PARAMETER(MXSDNPF=99,MXSDPRF=1,MXSDDPF=2)
  PARAMETER(MXSRNPF=1,MXSRPRF=1,MXSRDPF=1,MXSRMAF=1,MXSRESF=1)
C
C ----- 3. FOR TRANSPORT SOURCE/SINK AND BOUNDARY CONDITIONS
C
  PARAMETER(MXSELTP=1,MXSSPRT=1,MXSSDPT=1,MXSWNPT=1,MXSWPRT=1,
>    MXSWDPT=1)
  PARAMETER(MXSCNPT=1,MXSCEST=1,MXSCPRT=1,MXSCDPT=1)
  PARAMETER(MXSNNPT=1,MXSNEST=1,MXSNPRT=1,MXSNDPT=1)
  PARAMETER(MXSVPNT=1,MXSVEST=1,MXSVPRT=1,MXSVDPNT=1)
  PARAMETER(MXSDNPT=1,MXSDPRT=1,MXSDDPT=1)
C
  PARAMETER(MXSNPW=1,MXSELW=1)
C
C ===== MAXIMUM DIMENSION BLOCK FOR 1-D CANAL FLOW
C
  PARAMETER(MXCNRN=2,MXCNR=1)
  PARAMETER(MXCEL=8,MXCNP=9)
  PARAMETER(MXCCPM=1)
  PARAMETER(MXCGPR=1,MXCGPD=9)
  PARAMETER(MXCWPR=1,MXCWPD=1)
  PARAMETER(MXCRPR=2,MXCRPD=4)
  PARAMETER(MXCPPR=1,MXCPPD=4)
  PARAMETER(MXCUPR=1,MXCUPD=4)
  PARAMETER(MXCDPR=1,MXCDPD=4)
  PARAMETER(MXCNTC=400)
  PARAMETER(MXCNRJ=2,MXCANG=1)
  PARAMETER(MXCHMXR=1,MXCHMXD=4)
  PARAMETER(MXCHMNR=1,MXCHMND=4)
  PARAMETER(MXCSNP=12,MXCSEL=10)
  PARAMETER(MXCNGO=2)
C
C ===== MAXIMUM DIMENSION BLOCK FOR RETENTION POND
C
  PARAMETER(MXPONK=1,MXPPRK=1,MXDPPK=4,MXPOBK=30,MXOULK=6)
  PARAMETER(MXFXPK=1,MXFXDK=101)
C

```

C===== MAXIMUM DIMENSION BLOCK FOR 2-D OVERLAND FLOW

C

```

PARAMETER(MXOVTK=12,MXONTI=200,MXODTC=1)
PARAMETER(MXONP=99,MXOEL=80,MXOBNP=52,MXOBES=52)
PARAMETER(MXOHBP=12,MXOJBD=9,MXOKBD=6)
PARAMETER(MXORES=MXOEL,MXORPR=2,MXORDP=4)
PARAMETER(MXODNP=1,MXODPR=1,MXODDP=2)
PARAMETER(MXOCNP=1,MXOCES=1,MXOCPR=1,MXOCDP=4)
PARAMETER(MXONNP=5,MXONES=4,MXONPR=1,MXONDP=15)
PARAMETER(MXOMAT=1)

```

C

C===== MAXIMUM DIMENSION BLOCK FOR GRAPHIC INTERFACE

C

```

PARAMETER(MXXYPK=101,MXXYSK=7)
PARAMETER(MXPNTK=10,MXLINK=5,MXITPK=30)
PARAMETER(MXPSTK=100)

```

With the above descriptions, the input data can be prepared according to the instructions in Appendix A. To save space, the input data files and output results are given in the attached floppy disk. Figure 4.24 is the simulated water stages of channel and retention pond. Figure 4.25 and Figure 4.26 show the water stage and velocity distribution of 2-D overland flow at $t = 18,000$ sec. Figure 4.27 is the simulated total head contour at $z = 196$ m of 3-D subsurface domain.

In this example, 1-D canal is operated by the control structures designed to maintain the water stage in a range ($201 \text{ m} < H < 203 \text{ m}$) while interacting with the subsurface through infiltration. Because the upstream boundary gates are always open, the inflow flux rate is a constant value of $20 \text{ m}^3/\text{sec}$. For simplifying the estimation, the cross sections of all canal reaches are assumed rectangular with a bottom width of 4 m (given). The total length of canal is 400 m and the initial water stage is 202 m. The maximum volume of water that canal needs to reach 203 m of water stage is $1,600 (=400 \times 4 \times 1) \text{ m}^3$. During the first subsurface time step ($t = 1,800$ sec), $36,000 (=20 \times 1,800) \text{ m}^3$ of water flows into the canal. This amount of water is much greater than that the canal can take. As designed, the downstream boundary gates are opened only when the water stage is greater than 203 m. Therefore, the simulated water stage of the canal is slightly varied around 203 m as shown in Figure 4.24.

Since the volume of the inflow is larger than the canal capacity, the excess water is designed to flow out

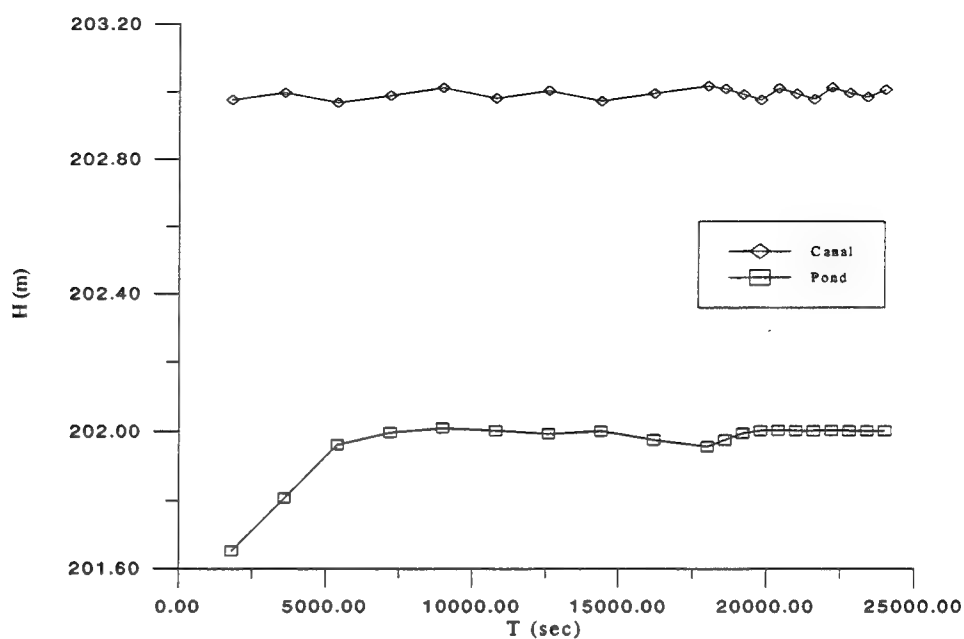


Figure 4.24 Simulated water stage of canal and retention pond

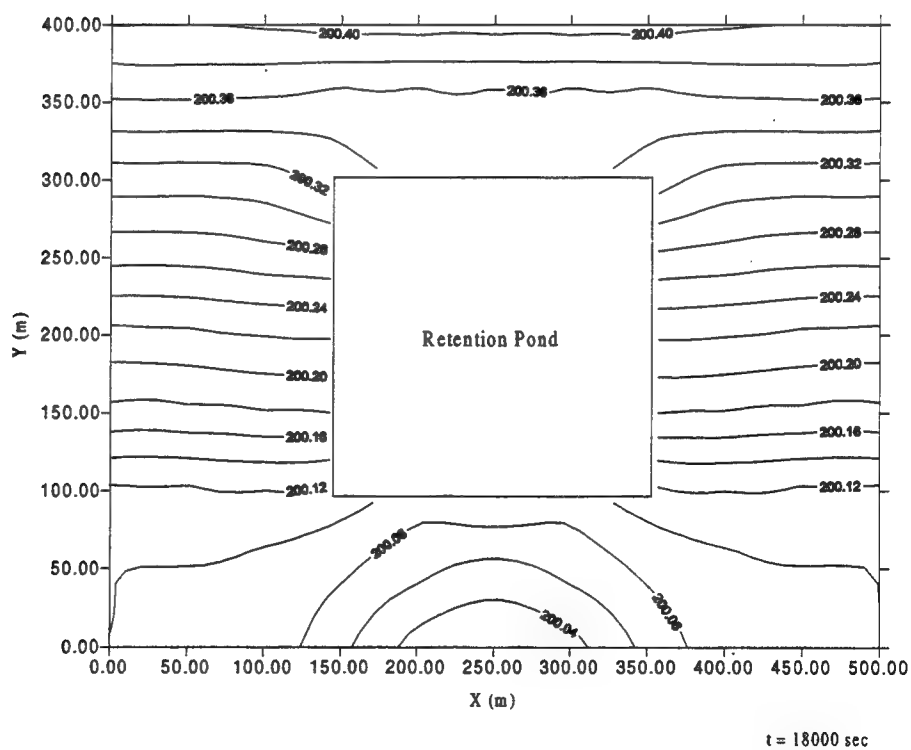


Figure 4.25 Simulated water stage of 2-D overland flow for Example 5

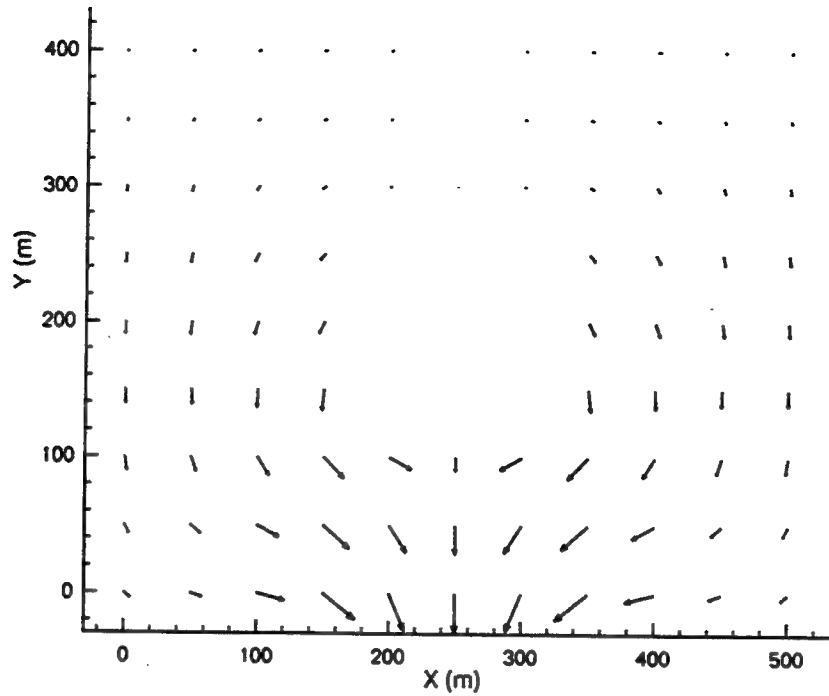


Figure 4.26 Simulated velocity field of 2-D overland flow at $t = 18000$ sec for Example 5

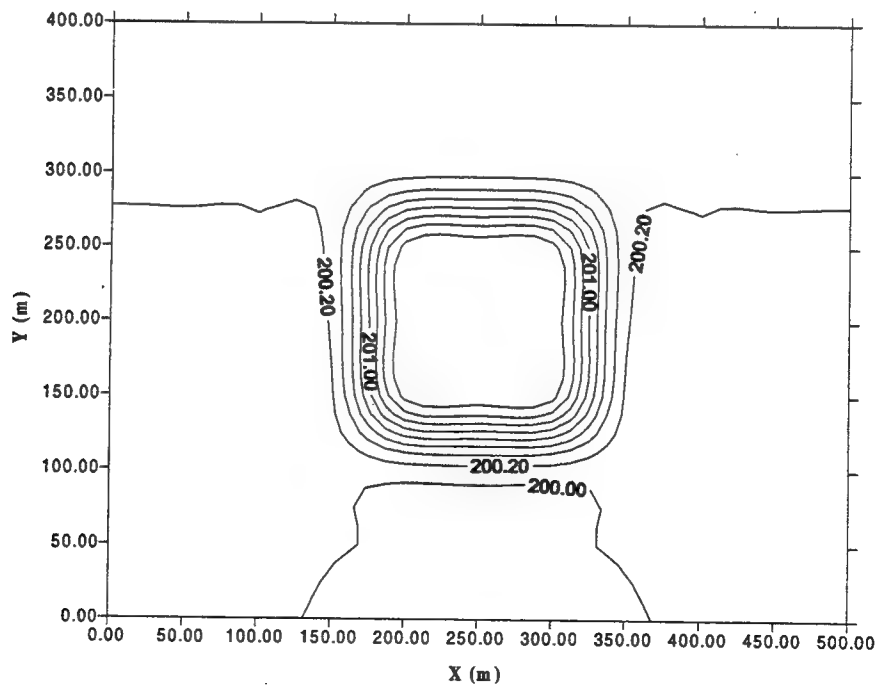


Figure 4.27 Total head contour at $Z = 196$ m of 3-D subsurface at $t = 18,000$ sec for Example 5

of the canal through the downstream gate and pump to the retention pond. From the diagnostic result of the canal at $t = 1,800$ sec in the output file, $20 \text{ m}^3/\text{sec}$ of water flows into the canal by upstream gate, $18 \text{ m}^3/\text{sec}$. The total volume of water pumped to retention pond is $1,620 (= 0.9 \times 1,800) \text{ m}^3$ and the bottom area of the retention pond is $10,000 (= 100 \times 100) \text{ m}^2$. The water stage of retention pond will be raised about $0.16 (= 1,620 / 10,000) \text{ m}$ to $201.66 (= 201.5 + 0.16) \text{ m}$. From the simulation results, the water stage of retention pond is, as expected, increased to 201.65 m at the first time step. From the continuous water recharge from canal, the simulated water stage of retention pond is gradually raised to 202 m . The excess amount of water is pumped to 2-D overland domain only when the water stage is higher than 202 m and the discharge rate is determined by a given rating curve. Therefore, as we predicted in the canal case, the simulated water stage of the retention pond is slightly varied around 202 m once it reaches 202 m (at $t = 9,000$ sec). Figure 4.24 demonstrates this result.

From the problem specification, we know that (1) the boundary condition for 2-D overland domain is impermeable except a radiation boundary condition is specified on the section between $(150 \text{ m}, 0 \text{ m})$ and $(350 \text{ m}, 0 \text{ m})$, (2) the rainfall is uniform, (3) the downhill bottom slope is $1/1,000$ from $y = 400 \text{ m}$ to $y = 0 \text{ m}$, and (4) the interaction between 2-D overland flow and 3-D subsurface flow is via infiltration. Because the retention pond is not included in the domain of 2-D simulation, it becomes an obstacle in the 2-D overland domain. Due to the designed downhill slope and the radiation downstream boundary, the water tends to flow toward the lower part of the domain. Owing to the existence of the retention pond in the 2-D overland domain, the flow is obstructed by the retention pond resulting in the higher stage and lower velocity in the upper area than in the bottom area of the domain. As shown in Figure 4.25, the retention pond is symmetric to the line across $x = 250 \text{ m}$, so is the water stage distribution.

From Figure 4.26, the velocity distribution is symmetric to the line across $x = 250 \text{ m}$ as the water stage distribution is. It is because of the specified boundary condition described above. The circulation pattern of velocity shown in Figure 4.26 is due to the exclusion of retention pond from the 2-D overland flow simulation.

The overland domain is discretized to 64 elements with a dimension of 50 m \times 50 m in each element (Figure 4.21). At $t = 18,000$ sec, the total volume of water falling on the 2-D overland is $57,600 \text{ m}^3 (= 2 \times 10^{-5} \times 64 \times 50 \times 50 \times 18,000)$. In the subsurface domain excluding the dike, there is a 4 m thickness of unsaturated zone with the moisture content ranging from 0.438 to 0.45 corresponding to the initial pressure head $h = -4$ m to 0 m. This part of unsaturated zone has a maximum void volume of $9,600 (= 4 \times 500 \times 400 \times 0.012) \text{ m}^3$. The initial condition of the dike has the moisture content ranging from 0.432 to 0.438 corresponding to the initial pressure head $h = -6$ m to -4 m. If the initial moisture content of dike is assumed an average value of 0.435 and the total volume of dike is calculated as 12 elements with a dimension of 2 m \times 50 m \times 50 m in each element (Figure 4.21), the unsaturated zone of the dike has an approximately void volume of $900 (= 12 \times 2 \times 50 \times 50 \times 0.015) \text{ m}^3$. Also, the total amount of water falling on the top of dike is $10,800 (= 2 \times 10^{-5} \times 12 \times 50 \times 50 \times 1,800) \text{ m}^3$. From the simulation results at $t = 18,000$ sec, the top layer of dike is still unsaturated which means the rainfall fully infiltrates into the subsurface domain and the flux type of variable boundary condition has been implemented on the top surface of dike. Therefore, the total amount of water from rainfall ($68,400 \text{ m}^3 = 57,600 \text{ m}^3 + 10,800 \text{ m}^3$) is much greater than the maximum void volume of unsaturated zone ($10,500 \text{ m}^3 = 9,600 \text{ m}^3 + 900 \text{ m}^3$) in the subsurface. At $t = 18,000$ sec, the subsurface domain is fully saturated except for on the very top part of the dike, and the retention pond is full of water as mentioned above. Figure 4.27 is the simulated total head distribution on the x-y cross section across $z = 196$ m at $t = 18,000$ sec.

Let us consider a system including the retention pond, 2-D overland domain, and 3-D subsurface domain. At $t = 18,000$ sec, the total amount of water pumped from the canal is $17,010 \text{ m}^3 (= 0.9 \times 1,800$ for the first time step $+ 9 \times 0.95 \times 1,800$ for the other 9 time steps: the number 0.95 is found in the diagnostic results of the output file). Also, the infiltration from the canal into the subsurface is negligible when compared to this amount of inflow water from the simulation results. The total amount of water from rainfall to the retention pond is $3,600 (= 2 \times 10^{-5} \times 4 \times 50 \times 50) \text{ m}^3$. Therefore, the total volume of water flows into this system is $89,010 (= 68,400 +$

17,010 + 3,600) m³.

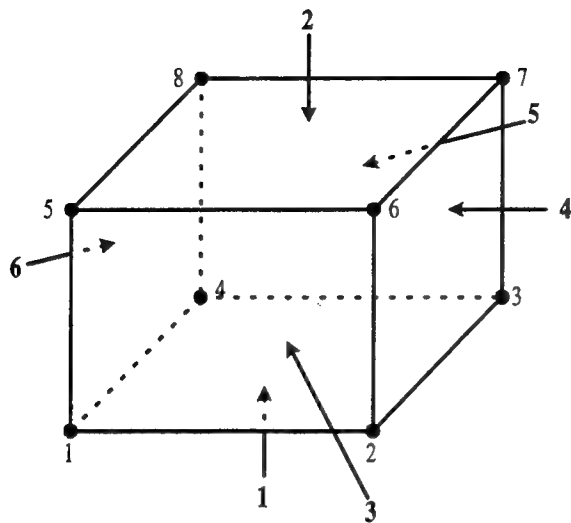
According to the simulation result at $t = 18,000$ sec, the accumulated amount of water flowing out of subsurface domain through the Dirichlet boundary is 41,411 m³, which accounts for 47% ($= 41,411 / 89,010 \times 100\%$) of inflow water. Meanwhile, the accumulated amount of water flowing out of overland domain through the radiation boundary is 16,300 m³, which accounts for 18% ($= 16,300 / 89,010 \times 100\%$) of inflow water. As the accumulation of water in the system, 592 m³ and 197 m³ are to increase the water content in the subsurface and on the overland regime, respectively. Also, the retention pond is full of water at $t = 18,000$ sec which accounts for 5,000 ($= 0.5 (= 202 - 201.5) \times 100 \times 100$) m³ of water. The accumulated amount of water increased in this system takes only 7% ($= (5,000 + 592 + 197) / 89,010 \times 100\%$) of inflow water. The imbalance of mass is due to numerical error. To reduce this error, two layers of small elements must be used to discretize the region along the global boundary.

APPENDIX A: DATA INPUT GUIDE FOR COSFLOW**A.1 Meshes**

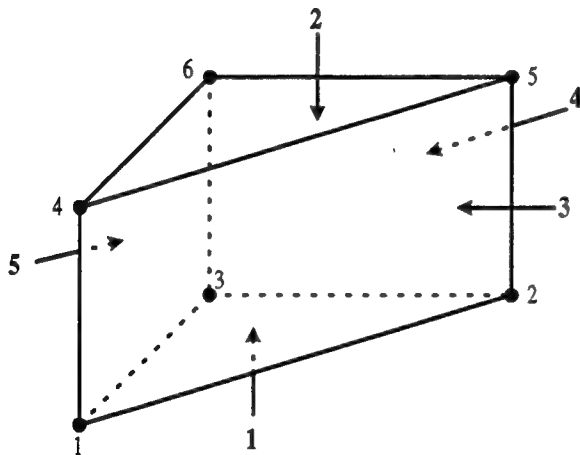
The computational discretization utilized by COSFLOW is a three-dimensional finite element mesh for 3-D subsurface and a two-dimensional finite element mesh for 2-D overland. The volumetric 3-D domain to be modeled by COSFLOW must be idealized and discretized into hexahedra, prisms, and/or tetrahedra, whereas 2-D region must be discretized to quadrilateral and/or triangular elements. Elements are typically grouped into zones representing different stratigraphic units. Each element is assigned a material ID representing the zone to which the elements belong. When constructing a mesh, care should be taken to ensure that elements do not cross or straddle stratigraphic boundaries.

The types of 3-D elements supported by COSFLOW are shown in Figure A.1. The numbering sequence and face ID shown in Figure A.1 should be used when describing the elements in the geometry file.

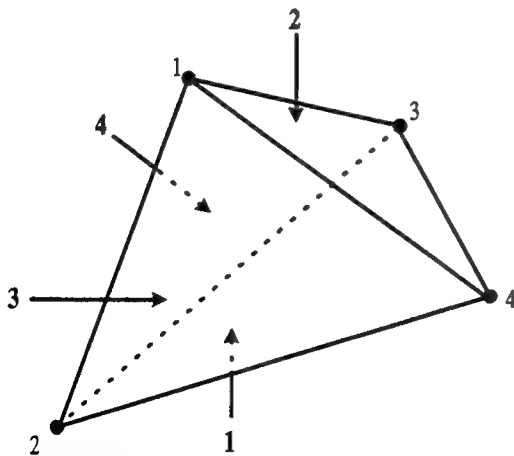
The coordinates of the mesh nodes and the element topology are input to COSFLOW through the geometry file. The format of the geometry file for 3-D elements is shown in Figure A.2. The cards used in the geometry file are specified in Table A.1.



Hexahedron	
Face	Nodes
1	1-4-3-2
2	5-6-7-8
3	1-2-6-5
4	2-3-7-6
5	3-4-8-7
6	4-1-5-8



Prism or Wedge	
Face	Nodes
1	1-3-2
2	4-5-6
3	1-2-5-4
4	2-3-6-5
5	3-1-4-6



Tetrahedron	
Face	Nodes
1	2-3-4
2	1-4-3
3	1-2-4
4	1-3-2

Figure A.1 Numbering sequence and face ID in a 3-D element

3DFEMGEO	/* File type identifier */
T1 text	/* Title, line 1 */
T2 text	/* Title, line 2 */
T3 text	/* Title, line 3 */
GN id x y z	/* Nodal coordinates */
GE8 id n1 n2 n3 n4 n5 n6 n7 n8 matid	/* Hexahedral element */
GE6 id n1 n2 n3 n4 n5 n6 matid	/* Prism element */
GE4 id n1 n2 n3 n4 matid	/* Tetrahedral element */
END	/* End of input data */

Figure A.2 The file format of 3-D geometry file

Table A.1 The data input format of geometry file for 3-D elements

<u>Value</u>		<u>Description</u>	
3DFEMGEO		File type identifier.	
<u>T1-T3 Cards</u>		<u>Job title</u>	
		Only one T1, T2, and T3 card can be used.	
<u>GN Card</u>		<u>Mesh node</u>	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	GN	Card group identifier Mesh node. One entry for each node in the mesh.
1	NP	+	The ID of the node.
2	X(NP,1)	±	The x coordinate of the node [L].
3	X(NP,2)	±	The y coordinate of the node [L].
4	X(NP,3)	±	The z coordinate of the node [L].
<u>GE Card</u>		<u>Mesh card</u>	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	GE	Card group identifier
C 3	IC3	8	Hexahedral element. One entry for each hexahedral element in the mesh. the ordering of the nodes should correspond to the diagram in Figure A.2
1	M	+	The ID of the element.
2	IE(M,1)	+	The 1 st global node number in this element.
3	IE(M,2)	+	The 2 nd global node number in this element.
4	IE(M,3)	+	The 3 rd global node number in this element.
5	IE(M,4)	+	The 4 th global node number in this element.
6	IE(M,5)	+	The 5 th global node number in this element.

7	IE(M,6)	+	The 6 th global node number in this element.
8	IE(M,7)	+	The 7 th global node number in this element.
9	IE(M,8)	+	The 8 th global node number in this element.
10	IE(M,9)	+	The material ID of this element.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	GE	Card group identifier
C 3	IC3	6	Wedge or prism element. One entry for each wedge element in the mesh. the ordering of the nodes should correspond to the diagram in Figure A.2

1	M	+	The ID of the element.
2	IE(M,1)	+	The 1 st global node number in this element.
3	IE(M,2)	+	The 2 nd global node number in this element.
4	IE(M,3)	+	The 3 rd global node number in this element.
5	IE(M,4)	+	The 4 th global node number in this element.
6	IE(M,5)	+	The 5 th global node number in this element.
7	IE(M,6)	+	The 6 th global node number in this element.
8	IE(M,9)	+	The material ID of this element.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	GE	Card group identifier
C 3	IC3	4	Tetrahedral element. One entry for each tetrahedral element in the mesh. the ordering of the nodes should correspond to the diagram in Figure A.2

1	M	+	The ID of the element.
2	IE(M,1)	+	The 1 st global node number in this element.
3	IE(M,2)	+	The 2 nd global node number in this element.
4	IE(M,3)	+	The 3 rd global node number in this element.
5	IE(M,4)	+	The 4 th global node number in this element.
10	IE(M,9)	+	The material ID of this element.

<u>EN Card</u>		<u>End card</u>	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	EN	Card group identifier
C 3	IC3	D	End of input data marker

The types of 2-D elements supported by COSFLOW are shown in Figure A.3. The numbering sequence shown in Figure A.3 should be used when describing the elements in the geometry file.

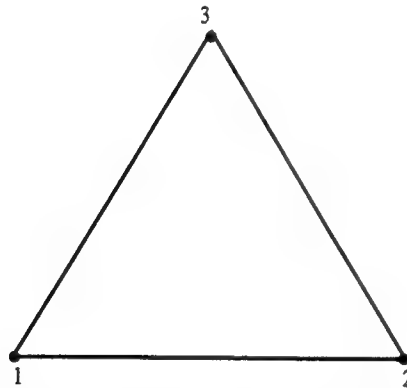
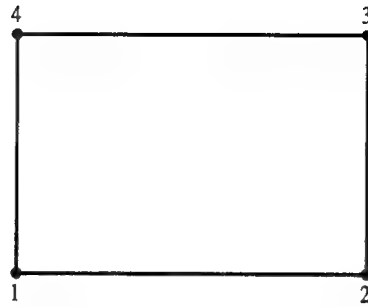


Figure A.3 Numbering sequence in a 2-D element

The coordinates of the mesh nodes and the element topology are input to COSFLOW through the geometry file. The format of the geometry file for 2-D elements is shown in Figure A.4.

MESH2D	/* File type identifier */
ND id x y z	/* Nodal coordinates */
E4Q id n1 n2 n3 n4 matid	/* Rectangular element */
E3T id n1 n2 n3 matid	/* Triangular element */
END	/* End of input data */

Figure A.4 Geometry file format for 2-D meshes

The cards used in the geometry file are as follows.

Table A.2 The data input format of geometry file for 2-D elements

<u>Value</u>		<u>Description</u>	
MESH2D		File type identifier.	
<u>ND Card</u>		<u>Mesh node</u>	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	ND	Card group identifier Mesh node. One entry for each node in the mesh.
1	NP	+	The ID of the node.
2	X(NP,1)	±	The x coordinate of the node [L].
3	X(NP,2)	±	The y coordinate of the node [L].
4	X(NP,3)	±	The z coordinate (elevation) of the node [L].
<u>E4 Card</u>		<u>Mesh card</u>	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	E4	Card group identifier
C 3	IC3	Q	Quadrilateral element. One entry for each rectangular element in the mesh. the ordering of the nodes should correspond to the diagram in Figure A.4
1	M	+	The ID of the element.
2	IE(M,1)	+	The 1 st global node number in this element.
3	IE(M,2)	+	The 2 nd global node number in this element.
4	IE(M,3)	+	The 3 rd global node number in this element.
5	IE(M,4)	+	The 4 th global node number in this element.
6	IE(M,5)	+	The material ID of this element.
<u>E3 Card</u>		<u>Mesh card</u>	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	E3	Card group identifier
C 3	IC3	T	Triangular element. One entry for each triangular element in the mesh. the ordering of the nodes should correspond to the diagram in Figure A.4
1	M	+	The ID of the element.
2	IE(M,1)	+	The 1 st global node number in this element.
3	IE(M,2)	+	The 2 nd global node number in this element.
4	IE(M,3)	+	The 3 rd global node number in this element.
5	IE(M,5)	+	The material ID of this element.

<u>EN Card</u>		<u>End card</u>	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	EN	Card group identifier
C 3	IC3	D	End of input data marker

A.2 Model file for 3-D subsurface flow and transport

The contents of this file consist of analysis options, material properties, boundary conditions, and initial conditions. The file type identifier and the title cards are similar to the corresponding cards found in the geometry file described in the previous section. The file content and format are shown in Table A.3.

Table A.3 The data input format of model file for 3-D subsurface flow and transport

<u>Value</u>		<u>Description</u>	
3DFEMWBC		File type identifier.	
<u>T1-T3 Cards</u>		<u>Job title</u>	
		Only one T1, T2, and T3 card can be used.	
<u>OP Card</u>		<u>Run option parameters</u>	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OP	Card group identifier
C 3	IC3	1	Specify the type of simulation;
1	KMOD	10	3-D subsurface flow simulation only,
		1	3-D transport only,
		11	3-D coupled flow and transport,
		100	Coupled 3-D subsurface and 1-D canal flows,
		110	Coupled 1-D canal, 2-D overland, and 3-D subsurface flows,
		20	2-D overland flow only;
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OP	Card group identifier
C 3	IC3	2	Solver options.
1	KSSF	0	Steady-state flow simulation,
		1	Transient flow simulation;
2	KSST	0	Steady-state transport simulation,
		1	Transient transport simulation;
3	ILUMP	0	No mass lumping,
		1	Mass lumping,

COSFLOW.DOC - 142

4	IMID	0	No mid-difference scheme is applied,
		1	Mid-difference scheme is applied,
5	IPNTSF/IPNTST		Matrix solver for flow and transport simulations.
		1	The pointwise iterative matrix solver
		2	Preconditioned Conjugate Gradient Method (polynomial),
		3	Preconditioned Conjugate Gradient Method (Incomplete Cholesky).
6	IQUAR		Index of using quadrature for numerical integration.
		11	Nodal quadrature for the integration of an element face and an element.
		12	Nodal quadrature for the integration of an element face, Gaussian quadrature for the integration of an element.
		21	Gaussian quadrature for the integration of an element face, Nodal quadrature for the integration of an element.
		22	Gaussian quadrature for the integration of an element face and an element.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OP	Card group identifier
C 3	IC3	3	Weighting factor for flow and transport simulations;
1	WF/WT		Time derivative weighting factor for flow and transport simulation,
		0.5	Crank-Nicolson central
		1.0	Backward difference
2	OMEF/OMET		Iteration parameter for solving nonlinear matrix equations.
		0.0-1.0	Under relaxation,
		1.0	Exact relaxation,
		1.0-2.0	Over relaxation.
3	OMIF/OMIT		Iteration parameter for solving linearized matrix equation pointwise.
		0.0-1.0	Under relaxation,
		1.0	Exact relaxation,
		1.0-2.0	Over relaxation.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OP	Card group identifier
C 3	IC3	4	Sorption option KSORP if KMOD = 1 or KMOD = 11; Otherwise, the index for analytical soil characteristic function KSP;
1	KSP	0	KSP = 0 is an index for providing an analytical soil characteristic function. The default value is 1.
		1	Linear isotherm,
		2	Freundlich isotherm,
		3	Langmuir isotherm;

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OP	Card group identifier
C 3	IC3	5	Parameters for Preconditioned Conjugate Gradient method;

1 GG Upper bound of the maximum eigenvalue of the coefficient matrix used in the Preconditioned Conjugate Gradient method.

IP CardIteration Parameters

Field	Variable	Value	Description
C 1-2	IC1	IP	Card group identifier
C 3	IC3	1	Iteration parameters for solving flow equations
1	NITERF	+	Number of iterations allowed for solving the non-linear flow equation.
2	NCYLF	+	Number of cycles permitted for iterating rainfall seepage boundary condition or the boundary condition on the interface between 2-D overland and 3-D subsurface regimes per time step.
3	NPITERF	+	Number of iterations permitted for solving the linearized flow equations, i.e., this value is used in matrix solver.
4	TOLAF	+	The convergent criteria for the implementation of iteration scheme in the steady-state flow simulation, (L).
5	TOLBF	+	The convergent criteria for the implementation of iteration scheme in the transient-state flow simulation, (L).

Field	Variable	Value	Description
C 1-2	IC1	IP	Card group identifier
C 3	IC3	2	Iteration parameters for solving transport equations
1	NITERT	+	Number of iterations allowed for solving the non-linear transport equation.
2	NPITERT	+	Number of iterations permitted for solving the linearized transport equations, i.e., this value is used in matrix solver.
3	TOLBT	+	The convergent criteria for the implementation of iteration scheme in the transient-state transport simulation, (L).

Field	Variable	Value	Description
C 1-2	IC1	IP	Card group identifier
C 3	IC3	3	Iteration parameters for either coupled flow and transport or coupled 1-D canal and 3-D subsurface flow simulation
1	NITFTT	+	Number of iterations allowed for solving the coupled nonlinear equations for transient solutions.
2	OMEFTT		Iteration parameter for solving the coupled nonlinear equations for transient solutions.
		0.0-1.0	Under relaxation,
		1.0	Exact relaxation,
		1.0-2.0	Over relaxation,

PT CardParticle Tracking Parameters

Field	Variable	Value	Description
C 1-2	IC1	PT	Card group identifier

C 3	IC3	1	Particle tracking parameters
1	NXW	+	The number of grids for sub-element tracking in the x-direction.
2	NYW	+	The number of grids for sub-element tracking in the y-direction.
3	NZW	+	The number of grids for sub-element tracking in the z-direction.
4	IDETQ		Index of particle tracking velocity.
		1	Average velocity of starting and ending points is used.
		2	Single velocity of starting point is used.

TC CardIteration Parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	TC	Card group identifier
C 3	IC3	1	Total simulation time.

1	TMAX	+	Maximum simulation time, (T)
---	------	---	------------------------------

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	TC	Card group identifier
C 3	IC3	2	Computational time step for 2-D overland flow only.
1	IDT		Time step type.
		0	Constant time-step size.
		1	Variable time-step size.
2	DELT	+	Time-step size is specified if IDT = 0.
	IDTXY	+	Index of xy series for variable time-step size if IDT = 1. The x values in the series represent times at which the time step-size will change to next y value. The y values represent the time step sizes.

OC CardOutput control parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OC	Card group identifier
C 3	IC3	1	Print control parameters.
1	IBUG	0	Do not print the diagnostic output,
		1	Do print the diagnostic output,
2	ICHNG	0	Do not print cyclic change of rainfall/seepage nodes or of ground surface nodes.
		1	Print cyclic change of rainfall/seepage nodes or of ground surface nodes.
3	JOPT	0	Print at specified interval,
		1	Print at specified set of time values,
4	KPRT	+	Specify the interval if JOPT = 0,
	NPRINT	+	Specify total number of the specified times, if JOPT=1,
(new line)		+	After OC1 card, specify the times, if JOPT = 1. One data entry per line.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OC	Card group identifier
C 3	IC3	2	Print options.
1	NSELT	+	Total options to be selected,
2	KPR0(i)		Print options,
		0	Print nothing
		1	Print flow information at the boundary
		2	Print total head
		3	Print pressure head
		4	Print concentration
		5	Print flux
		6	Print nodal moisture content
		7	Print Darcy velocity

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OC	Card group identifier
C 3	IC3	3	Output control for post-process.
1	IFILE	0	Save as ASCII test format,
		1	Save as binary format,
2	KOPT	0	Save at specified interval,
		1	Save at specified set of times,
3	KDSK	+	Save every specified interval, if KOPT=0,
	KPOST	+	Save at specified set of times, if KOPT=1,
(new line)		+	After OC3 card, specify the times, if KOPT = 1. One data entry per line, up to KPOST times.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OC	Card group identifier
C 3	IC3	4	Solution file output options.
1	KSELT	+	Total options to be selected,
2	KSAVE(i)		Save options,
		0	Save nothing
		1	Save pressure head
		3	Save nodal moisture content
		4	Save velocity
		5	Save concentration

MP CardMaterial property parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	MP	Card group identifier
C 3	IC3	1	Index of hydraulic conductivity or permeability
1	KCP	0	Hydraulic conductivity.

		1	Permeability.
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	MP	Card group identifier
C 3	IC3	2	Hydraulic conductivity or permeability tensor.
1	I	+	Material ID.
2	PROPF(1,I)	+	Saturated xx hydraulic conductivity, K_{xx} [L/T], or saturated permeability [L^2] of the medium I.
3	PROPF(2,I)	+	Saturated yy hydraulic conductivity, K_{yy} [L/T], or saturated permeability [L^2] of the medium I.
4	PROPF(3,I)	+	Saturated zz hydraulic conductivity K_{zz} [L/T], or saturated permeability [L^2] of the medium I.
5	PROPF(4,I)	+	Saturated xy hydraulic conductivity K_{xy} [L/T], or saturated permeability [L^2] of the medium I.
6	PROPF(5,I)	+	Saturated xz hydraulic conductivity K_{xz} [L/T], or saturated permeability [L^2] of the medium I.
7	PROPF(6,I)	+	Saturated yz hydraulic conductivity K_{yz} [L/T], or saturated permeability [L^2] of the medium I.
8	PROPF(7,I)	+	The modified compressibility of the medium I, α' [1/L].
9	PROPF(8,I)	+	The modified compressibility of the water, β' [1/L], in the medium I.
10	PROPF(9,I)	+	The effective porosity of the medium I, n_e [dimensionless].
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	MP	Card group identifier
C 3	IC3	3	Density and viscosity of fresh water.
1	RHO	+	Density of fresh water, [M/L ³]
2	VISC	+	Dynamic viscosity of water, [M/L/T]
3	GRAV	+	Acceleration of gravity, [L/T ²].
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	MP	Card group identifier
C 3	IC3	4	Concentration dependence coefficients. The relationship density and viscosity functions are: $\rho/\rho_0 = a_1 + a_2 C + a_3 C^2 + a_4 C^3$, $\mu/\mu_0 = a_5 + a_6 C + a_7 C^2 + a_8 C^3$,
1	RHOMU(1)	\pm	Coefficient a_1 .
2	RHOMU(2)	\pm	Coefficient a_2 . The units are the inverse of concentration units.
3	RHOMU(3)	\pm	Coefficient a_3 . The units are the inverse of concentration units.
4	RHOMU(4)	\pm	Coefficient a_4 . The units are the inverse of concentration units.
5	RHOMU(5)	\pm	Coefficient a_5 .
6	RHOMU(6)	\pm	Coefficient a_6 . The units are the inverse of concentration units.
7	RHOMU(7)	\pm	Coefficient a_7 . The units are the inverse of concentration units.
8	RHOMU(8)	\pm	Coefficient a_8 . The units are the inverse of concentration units.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	MP	Card group identifier
C 3	IC3	5	Dispersion/diffusion coefficients.
1	I	+	Material type ID.
2	PROPT(1,I)	+	Distribution coefficient, K_d , Freundlich K, or Langmuir K for medium I [L^3/M].
3	PROPT(2,I)	+	Bulk density, ρ_b for medium I [M/L^3].
4	PROPT(3,I)	+	Longitudinal dispersion, α_L for medium I, [L].
5	PROPT(4,I)	+	Transverse dispersion, α_T for medium I, [L].
6	PROPT(5,I)	+	Molecular diffusion coefficient, α_M for medium I, [L^2/T].
7	PROPT(6,I)	+	Tortuosity, τ for medium I, [dimensionless].
8	PROPT(7,I)	+	Decay constant, λ for medium I, [$1/T$].
9	PROPT(8,I)	+	Freundlich n or Langmuir S_{max} for medium I, [dimensionless].
10	PROPT(9,I)	+	The modified compressibility of the medium I, α' [$1/L$].
11	PROPT(10,I)	+	The effective porosity of the medium I, n_e [dimensionless].
12	PROPT(11,I)	+	The first order biodegradation rate constant through dissolved phase for medium I, K_w [$1/T$].
13	PROPT(12,I)	+	The first order biodegradation rate constant through absorbed phase for medium I, K_s [$1/T$].
14	PROPT(13,I)	+	The modified compressibility of the water, β' [$1/L$], in the medium I.

SP Card**Soil property parameters for unsaturated zone**

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	SP	Card group identifier
C 3	IC3	1	Dispersion/diffusion coefficients.
1	I	+	Material type ID.
2	IHM(I)	+	Index of moisture content vs. pressure head xy series.
3	IHC(I)	+	Index of relative hydraulic conductivity vs. pressure head xy series.
4	IHW(I)	+	Index of water capacity vs. pressure head xy series.

PS Card**Point source/sink parameters**

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	PS	Card group identifier
C 3	IC3	1	Point source/sink flow rate for both flow and transport simulation.

1	NPWF(I)	+	Node number of the I-th point source/sink,
2	IWTYPF(I)	+	Index of the flow rate xy series.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	PS	Card group identifier
C 3	IC3	2	Point source/sink concentration for transport simulation.

1	NPWT(I)	+	Node number of the I-th point source/sink,
---	---------	---	--

2 IWTYPT(I) + Index of the concentration xy series.

DB Card Dirichlet boundary parameters

Field	Variable	Value	Description
C 1-2	IC1	DB	Card group identifier
C 3	IC3	1	Dirichlet boundary conditions, prescribed total head for flow simulation.

1 NPDBF(I) + Global node number of the I-th Dirichlet boundary,
 2 IDTYPF(I) + Index of the head xy series.

Field	Variable	Value	Description
C 1-2	IC1	DB	Card group identifier
C 3	IC3	2	Dirichlet boundary conditions, prescribed concentration for transport simulation.

1 NPDBT(I) + Global node number of the I-th Dirichlet boundary,
 2 IDTYPT(I) + Index of the concentration xy series.

CB Card Cauchy boundary parameters

Field	Variable	Value	Description
C 1-2	IC1	CB	Card group identifier
C 3	IC3	1	Cauchy boundary conditions, prescribed flow flux rate for flow simulation.

1 M + Element number of the Cauchy boundary,
 2 IDCF(I) + Element face ID of this boundary side,
 3 ICTYPF(I) + Index of the flux rate xy series.

Field	Variable	Value	Description
C 1-2	IC1	CB	Card group identifier
C 3	IC3	2	Cauchy boundary conditions, prescribed mass flux rate for transport simulation.

1 M + Element number of the Cauchy boundary,
 2 IDCT(I) + Element face ID of this boundary side,
 3 ICTYPT(I) + Index of the mass flux rate xy series.

NB Card Neumann boundary parameters

Field	Variable	Value	Description
C 1-2	IC1	NB	Card group identifier
C 3	IC3	1	Neumann boundary conditions, prescribed flow flux rate (due to pressure head difference) for flow simulation.

1 M + Element number of the Neumann boundary,
 2 IDNF(I) + Element face ID of this boundary side,

3	INTYPF(I)	+	Index of the flux rate xy series.
---	-----------	---	-----------------------------------

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	NB	Card group identifier
C 3	IC3	2	Neumann boundary conditions, prescribed dispersive mass flux rate for transport simulation.

1	M	+	Element number of the Neumann boundary,
2	IDNT(I)	+	Element face ID of this boundary side,
3	INTYPT(I)	+	Index of the dispersive mass flux rate xy series.

RS Card Variable boundary parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	RS	Card group identifier
C 3	IC3	1	Variable boundary condition, rainfall/evaporation/seepage flux rate for flow simulation

1	M	+	Element number of the rainfall/seepage boundary,
2	IDRF(I)	+	Element face ID of this boundary side,
3	IVTYPF(I)	+	Index number of rainfall/evaporation time series for the element face.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	RS	Card group identifier
C 3	IC3	2	Variable boundary conditions, prescribed concentration for transport simulation.

1	M	+	Element number of the Variable boundary,
2	IDRT(I)	+	Element face ID of this boundary side,
3	IVTYPT(I)	+	Index number of concentration time series for the element face.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	RS	Card group identifier
C 3	IC3	3	Variable boundary condition, ponding depth and minimum pressure head.

1	HCON	+	The ponding depth.
2	HMIN	±	The minimum pressure head.

IC Card Initial condition data parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IC	Card group identifier
C 3	IC3	S	Initial condition start type.

1	ISTART	0	Cold start.
		1	Hot start.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IC	Card group identifier
C 3	IC3	T	Initial condition hot start time.

1	HSTIME	+	Time corresponding for the hot start
---	--------	---	--------------------------------------

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IC	Card group identifier
C 3	IC3	H	Constant or variable initial total head.
1	IHEAD	0	Constant total head.
		1	Spatially variable total head.
2	HCONST	±	If IHEAD=0, the total head value. Otherwise, omit.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IC	Card group identifier
C 3	IC3	C	Constant or variable initial concentration.
1	ICON	0	Constant initial concentration.
		1	Spatially variable initial concentration.
2	CONVAL	±	If ICON=0, the concentration value. Otherwise, omit.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IC	Card group identifier
C 3	IC3	F	Initial condition file format.
1	ICFILE	0	The initial condition files are in text format.
		1	The initial condition files are in binary format.
2	IVFILE	0	The initial flow files are in text format.
		1	The initial flow files are in binary format.
3	IFFU	0	Flow files contain a steady-state solution.
		1	Flow files contain a transient solution.

XY Card**X-Y Series Parameters**

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	XY	Card group identifier
C 3	IC3	1	Generation of any x-y series function.
1	i	+	Index number for x-y series,
2	NPOINT(i)	+	The number of x-y values in the series.
3	DELTA X	0	Dummy values,
4	DELTA Y	0	Dummy values,
5	REPEAT	0	Dummy values,
6	BEGCYC	0	Dummy values,
7	TNAME	+	A character string representing the name of the XY series.
(new line)X,Y		±, ±	After XY card, the x-y values of the series are listed one pair per line

up to NPOINT(i).

<u>EN Card</u>		<u>End of data control</u>	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C1-2	IC1	EN	Card group identifier
C 3	IC3	D	End of input data

A.3 Model file for 1-D canal flow

Table A.4 lists the data format providing the required information for the simulation of 1-D canal flow.

Table A.4 The data input format of model file for 1-D canal flow simulation

The following cards are used to define the canal reach:

<u>Value</u>		<u>Description</u>	
CANAL		File type identifier.	

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	CR	Card group identifier
C 3	IC3	N	Groundwater nodes making up the canal reach;
1	IREACH(i)	+	The id of the reach
2	NPIRCH(i)	+	The number of canal nodes in the reach,
(new line)	IRN(i, j)	+	The associated groundwater node id of the canal nodes. They should be listed in upstream to downstream order.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	CR	Card group identifier
C 3	IC3	A	Geometric attributes of a canal reach
1	NCELM(i)	+	The id of the reach
2	RT1(i)	+	The elevation below the sediment of the reach,
3	RT2(i)	+	The top (canal bank) elevation of the reach,
4	RT3(i)	+	The bottom width of the reach,
5	RT4(i)	+	The side slope of the reach,
6	RT5(i)	+	The thickness of the canal bed material for the reach
7	RT6(i)	+	The hydraulic conductivity of the bed material,

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	CR	Card group identifier
C 3	IC3	W	Water stage attributes of a canal reach

1	id	+	The id of the reach
2	Ixymin	+	The id of the xy series describing time vs. the minimum water stage
3	Ixymax	+	The id of the xy series describing time vs. the maximum water stage
4	H RTP(id)	+	The initial water stage of the reach

The following cards are used to define the canal reach node:

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IF	Card group identifier
C 3	IC3	S	Index of simulating the interface between 1-D canal and 3-D subsurface system
1	IFACE	+	1 = the fluxes through the interface are treated as line sources/sinks to the 3-D subsurface system (narrow and shallow canal), 2 = the fluxes through the interface are treated as vertical plane sources/sinks to the 3-D subsurface system (narrow but deep channel) The default value is 2.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	CN	Card group identifier
C 3	IC3	S	Control structures at the reach node
1	id	+	The id of the reach node,
2	IDNPRH(id)		Type of structure,
		1	Interior gate
		2	Weir
		3	Upstream boundary gate
		4	Downstream boundary gate
		5	Structure-free joint
		6	Dead end
3	ICSON(id)	0	The gate is closed initially
		1	The gate is open initially
			Note: 1 is given if IDNPRH(id) = 2, 5, or 6.
4	IDPRS	+	The id of the submerged rating curve xy series (Q vs. ΔH) for the control structure being input
			Note: any integer can be given if IDNPRH(id) = 3.
5	IDPRN	+	The id of the non-submerged rating curve xy series (Q vs. h) for the control structure being input.
			Note: the rating curve is Q vs. t if IDNPRH(id) = 3.
6	IDPRT	+	The id of stage-prescribed curve xy series (t vs. H) for the tailwater of downstream boundary gate (i.e., for the case IDNPRH(id) = 4).
			Note: 1 is given for the other cases.
7	WBE(id)	+	The bottom elevation of the gate/weir being input.
8	WCD(id)	+	The opening height of the gate/weir being input.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	CN	Card group identifier
C 3	IC3	P	Pumping station at a reach node;
1	IDCMP(i)	+	The pump is located at reach node i
2	NPGPMP(1,i)	0	Pump to another reach,
		1	Pump to a groundwater node
		2	Pump to a retention pond
3	NPGPMP(2,i)	+	The id of the object (reach node, subsurface node, or retention pond) that the water is being pumped to
4	IPPON(i)	0	The pump is OFF initially.
		1	The pump is ON initially.
5	Ixy	+	The id of the xy series defining time vs. Q.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	CN	Card group identifier
C 3	IC3	N	Siphon station at a reach node;
1	id	+	The siphon is located at the reach node
2	Ixy	+	The id of the xy series defining Q vs. ΔH

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	CM	Card group identifier
C3	IC3	H	Convergent criterion,
1	DHRT	+	Allowed reach head changes in one time step,

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C1-2	IC1	CH	The i identifier
C 3	IC3	C	The head change criterion for convergence;
1	TOLBC	+	Convergent criterion for head

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	CT	Card group identifier
C 3	IC3	S	Time factor for canal time step,
1	NGWTI	+	The subdivision factor,

Additional cards are needed but not implement in GMS

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	CJ	Card group identifier
C 3	IC3	N	Canal Junctions;

COSFLOW.DOC - 154

1	JUNC(i)	+	Canal junction number i,
2	NJTRH(i)	+	Number of reaches connect to the i-th junction,
(new line)	IJTRH(j)	+	Reaches joint at the i-th junction,

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	PR	Card group identifier
C 3	IC3	1	Precipitation Data in routing reaches;

1	IREACH(i)	+	The id of the reach
2	ITPPRH(i)	+	The id of the xy series for precipitation,

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	AN	Card group identifier
C 3	IC3	G	Operation controlled by observation wells;

1	IFA	1	Controlled by Angel's well.
		2	Controlled by any specified head series .
2	JANGEL	+	Groundwater node at Angel's well if IFA=1
	Ixy	+	The id of the xy series defining time vs. head if IFA=2

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	SC	Card group identifier
C 3	IC3	1	Special control at S-197 for opening ;

1	IN197	+	Reach node at S-197
2	IR177O	+	Reach controlled the S-197
3	IR18CO	+	Reach controlled the S-197
4	OPEL11	+	Control elev. at the higher level at the S-197
5	OPEL12	+	Control elev. at the middle level at the S-197
6	OPEL13	+	Control elev. at the lower level at the S-197
7	OPEL21	+	Control elev. at the higher level at the S-18C
8	OPEL22	+	Control elev. at the middle level at the S-18C
9	OPEL23	+	Control elev. at the lower level at the S-18C

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	SC	Card group identifier
C 3	IC3	2	Special control at S-197 for closing ;

1	IN197	+	Reach node at S-197
2	IR176C	+	Reach controlled the S-197
3	IR177C	+	Reach controlled the S-197
4	IR197C	+	Reach controlled the S-197
5	CLEL1	+	Control elevation at the IR176C
6	CLEL2	+	Control elevation at the IR177C
7	CLEL3	+	Control elevation at the IR197C

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	SC	Card group identifier
C 3	IC3	3	Special control at S-332 pumping station ;
1	IR175	+	Reach id at S-332
2	EL175	+	Control elevation at the S-332
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	SC	Card group identifier
C 3	IC3	4	Special control at S-180 pumping station ;
1	IR180	+	Reach id at S-180
2	EL18CP	+	Control elevation at the S-18C
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	SP	Card group identifier
C 3	IC3	1	Special control at S-331 for Angel's well ;
1	IR331	+	Reach id at S-331
2	AEL1	+	Control elev. of upper level at Angel's well
3	AEL2	+	Control elev. at lower level at Angel's well
4	E1_331	+	Control elevation of upper level at S-331
5	E2_331	+	Control elevation of lower level at S-331
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	SP	Card group identifier
C 3	IC3	2	Special control at S-331 for flood control ;
1	IR331	+	Reach id at S-331
2	IR176	+	Reach no. at S-176
3	IR177	+	Reach no. at S-177
4	IR18C	+	Reach no. at S-18C
5	EL176	+	Control elevation at S-176
6	EL177	+	Control elevation at S-177
7	EL18C	+	Control elevation at S-18C
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	SP	Card group identifier
C 3	IC3	3	Special control at S-331 water supply
1	IN331	+	Reach node at S-331
2	IR176W	+	Reach controlled the S-331
3	IR177W	+	Reach controlled the S-331
4	IR196W	+	Reach controlled the S-331
5	IR194W	+	Reach controlled the S-331
6	IR332W	+	Reach controlled the S-331

COSFLOW.DOC - 156

7	EWS176	+	Control elevation at S-176
8	EWS177	+	Control elevation at S-177
9	EWS196	+	Control elevation at S-196
10	EWS194	+	Control elevation at S-194
11	EWS332	+	Control elevation at S-332

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OC	Card group identifier
C 3	IC3	5	Output control options
1	JSELT	+	Number of save options to be selected
2	JSAVE(i)	1	save water stage at the reach,
		2	Save net-flux for the reach,
		3	Save upstream flux,
		4	Save downstream flux,
		5	Save flux from subsurface flow
		6	Save flux from pumping,
		7	Save flux from rainfall

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OP	Card group identifier
C 3	IC3		Output control options
1	ICASE	1	Selection of C-111 operation,
		2	selection of new operation at triangle land

<u>EN Card</u>		<u>End of data control</u>	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C1-2	IC1	EN	Card group identifier
C 3	IC3	D	End of input data

A.4 Model file for 2-D overland flow and retention pond

Table 4.5 lists the data format providing the required information for the simulation of 2-D overland flow.

Table 4.6 lists the data input guide for retention pond.

Table A.5 The data input format of model file for 2-D overland flow simulation

<u>Value</u>	<u>Description</u>
2DWMSWBC	File type identifier.

T1-T3 Cards**Job title**

Only one T1, T2, and T3 card can be used.

OP Card**Run option parameters**

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OP	Card group identifier
C 3	IC3	1	Solver specification;
1	ILUMP	0	No mass lumping,
		1	Mass lumping,
2	IMID	0	No mid-difference scheme is applied,
		1	Mid-difference scheme is applied,
3	IPNTS		Matrix solver for flow simulations.
		0	Direct solver
		1	The pointwise iterative matrix solver
		2	Preconditioned Conjugate Gradient Method (polynomial),
		3	Preconditioned Conjugate Gradient Method (Incomplete Cholesky).
4	IVML		Option for matrix composition for velocity calculation.
		0	No mass lumping is applied,
		1	Mass lumping and direct solver are applied to compute velocity.
5	IQUAR		Index of using quadrature for numerical integration.
		11	Nodal quadrature for line and element integration
		12	Nodal quadrature for line integration, Gaussian quadrature for element integration
		21	Gaussian quadrature for line integration, Nodal quadrature for element integration,
		22	Gaussian quadrature for line and element integration.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OP	Card group identifier
C 3	IC3	2	Weighting factor for simulations;
1	W		Time derivative weighting factor for flow simulation,
		0.5	Crank-Nicolson central
		1.0	Backward difference
2	OME		Iteration parameter for solving nonlinear matrix equations.
		0.0-1.0	Under relaxation,
		1.0	Exact relaxation,
		1.0-2.0	Over relaxation,
3	OMI		Iteration parameter for solving linearized matrix equation pointwise.
		0.0-1.0	Under relaxation,
		1.0	Exact relaxation,
		1.0-2.0	Over relaxation
4	VMIN	+	Minimum velocity magnitude. It is considered numerical error below

this value.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OP	Card group identifier
C 3	IC3	3	Parameters for Preconditioned Conjugate Gradient method;
1	IEIGEN		Signal of parameter estimation for GG in the polynomial preconditioned conjugate gradient method;
		0	GG will not be read
		1	GG will be read
2	GG	+	Upper bound of the maximum eigenvalue of the coefficient matrix used in the Preconditioned Conjugate Gradient method.

IP Card**Iteration Parameters**

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IP	Card group identifier
C 3	IC3	1	Iteration parameters for solving flow equations
1	NITER	+	Number of iterations allowed for solving the non-linear flow equation.
2	NPITER	+	Number of iterations permitted for solving the linearized flow equations when IPNTS is set to 0 or 1.
3	TOLB	+	The convergent criteria for the implementation of iteration scheme.

TC Card**Time Control Parameters**

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	TC	Card group identifier
C 3	IC3	1	Total simulation time.
1	TMAX	+	Maximum simulation time, (T)
2	DELT	+	Initial time step size

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	TC	Card group identifier
C 3	IC3	2	Computational time step for 2-D overland flow only.
1	IDTXY	+	Index of xy series for variable time step,

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	TC	Card group identifier
C 3	IC3	3	Computational time step for coupled 1-D, 2-D, and 3-D flow simulation.
1	NCTI	+	Number of canal time steps is composed of a 2-D overland time step, if 1-D canal flow is simulated. A subdivision factor is provided for the preparation of overland time step, if 1-D canal flow is not involved.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	TC	Card group identifier
C 3	IC3	4	Time unit and length unit
1	IUNIT	+	A number is provided to indicate the time unit in the simulation. 0 = second, 1 = minute, 2 = hour, 3 = day, 4 = month, 5 = year.
2	LUNIT	+	A number is provided to indicate the length unit in the simulation. 0 = meter, 1 = Km, 2 = cm, 3 = ft, 4 = mile.

OC CardOutput control parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OC	Card group identifier
C 3	IC3	1	Print control parameters.
1	IBUG	0	Do not print the diagnostic output,
		1	Do print the diagnostic output,
2	JOPT	0	Print at specified interval,
		1	Print at specified set of time values.
3	KPRT	+	Specify the interval if JOPT = 0,
	NPRINT	+	Specify total number of the specified times, if JOPT=1,
(new line)		+	After OC1 card, specify the times, if JOPT = 1. One data entry per line.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OC	Card group identifier
C 3	IC3	2	Print options.
1	NSELT	+	Total options to be selected,
2	KPR0(i)		Print options,
		1	Mass balance table,
		2	Print water depth,
		3	Print water stage,
		4	Print velocity.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OC	Card group identifier
C 3	IC3	3	Output control for post-process.
1	IFILE	0	Save as ascii format,
		1	Save as binary format,
2	KOPT	0	Save at specified interval,
		1	Save at specified set of times,
3	KDSK	+	Save every specified interval, if KOPT=0,
	NPOST	+	Save at specified set of times, if KOPT=1,
(new line)		+	After OC3 card, specify the times, if KOPT = 1. One data entry per line.

COSFLOW.DOC - 160

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OC	Card group identifier
C 3	IC3	4	Solution file output options.

1	KSELT	+	Total options to be selected,
2	KSAVE(i)		Save options,
		1	Save water depth,
		2	Save water stage,
		3	Save flux,
		4	Save infiltration rate.

GC Card

Cross section parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	GC	Card group identifier
C 3	IC3	1	Generate cross section to calculate discharge.

1	NGCL	+	Total number of cross sections
---	------	---	--------------------------------

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	GC	Card group identifier
C 3	IC3	2	Cross section information.
1	id	+	Index number of cross section.
2	NPL	+	Number of nodal points in the x-section.
(new line)		+	Node points

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	GC	Card group identifier
C 3	IC3	3	Save and print hydrograph.
1	LOPT	0	Save and print at specified interval,
		1	Save and print at a set of times,
2	LDSK	+	Save and print at specified interval, if LOPT=0,
	LPOST	+	number of points is specified in the following set of times, if LOPT=1,
(new line)		+	After GC3 card, specify the times, if LOPT = 1.
			One data entry per line.

MP Card

Material property parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	MP	Card group identifier
C 3	IC3	1	Material property.
1	i	+	Material type id.
2	PROP(1,i)	+	Manning's roughness,

RF Card

Rainfall parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	RF	Card group identifier
C 3	IC3	1	Rainfall data for the simulation. This card can be eliminated if there is no rainfall all the time.

1	i	+	Element number at the surface boundary,
2	IRTYP(i)	+	Index of rainfall xy series for the element.

HB CardDepth boundary parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	HB	Card group identifier
C 3	IC3	1	Depth time series for the simulation;

1	NPDB(i)	+	i-th global node for head boundary,
2	IDTYP(i)	+	Index no. of water depth time series at the i-th node.

FB CardFlux boundary parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	FB	Card group identifier
C 3	IC3	1	Flux boundary information.
1	M	+	Element number on flux boundary,
2	ISC(3,i)	+	First global node on the i-th boundary segment,
3	ISC(4,i)	+	Second global node on the i-th boundary segment,
4	ICTYP(i)	+	Index of xy series for normal flux through the i-th flux boundary.

RB CardFlux boundary parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	RB	Card group identifier
C 3	IC3	1	Radiation boundary information.
1	M	+	Element number on radiation boundary,
2	ISN(3,i)	+	First global node on the i-th boundary segment,
3	ISN(4,i)	+	Second global node on the i-th boundary segment,
4	INTYP(i)	+	Index of xy series describing the rating curve for normal flux through the i-th radiation boundary.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	MN	Card group identifier
C 3	IC3	1	Minimum pressure head must be kept for each ground surface node.
1	IHMIN	0	Constant minimum pressure head
		1	Variable depth, (Not available)
2	HMIN	+	Value of the constant minimum pressure head.

IC CardInitial condition data parameters

COSFLOW.DOC - 162

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IC	Card group identifier
C 3	IC3	H	Water depth;
1	IHEAD	0	Constant depth,
		1	Variable depth,
2	HCONST	+	Value of the constant depth.

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IC	Card group identifier
C 3	IC3	S	Initial condition start type.
1	ISTART	0	Cold start, water depth and velocity
		1	Hot start, water depth and velocity

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IC	Card group identifier
C 3	IC3	T	Initial condition start time.
1	HSTIME	+	Time corresponding for the hot start

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IC	Card group identifier
C 3	IC3	F	Initial condition start type.
1	ICFILE	0	Text format
		1	Binary format

XY Card

X-Y Series Parameters

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	XY	Card group identifier
C 3	IC3	1	Generation of any x-y series function.
1	i	+	Index number for x-y series,
2	NPOINT(i)	+	The number of x-y values in the series.
3	DELTAX	0	Dummy values,
4	DELTAY	0	Dummy values,
5	REPEAT	0	Dummy values,
6	BEGCYC	0	Dummy values,
7	TNAME	+	A character string representing the name of the XY series.
(new line)X,Y			After XY card, the x-y values of the series are listed one pair per line up to NPOINT(i).

EN Card

End of data control

<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C1-2	IC1	EN	Card group identifier

C 3 IC3 D End of input data

Table A.6 The data input format of model file for 2-D retention pond simulation

<u>Value</u>		<u>Description</u>	
2DRPONBC		File type identifier.	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OP	Card group identifier
C 3	IC3	I	Define the retention ponds
1	NRPOND	+	Number of retention ponds
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IC	Card group identifier
C 3	IC3	H	Initial water surface elevation of each retention pond
1	ID	+	The id of the retention pond,
2	HR(ID)	+	The initial water surface elevation of the retention pond.
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	MX	Card group identifier
C 3	IC3	1	Maximum water surface elevation below the spillway for each retention pond
1	ID	+	The id of the retention pond
2	HMXPON(ID)	+	The maximum water surface elevation below the spillway of the retention pond.
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	MX	Card group identifier
C 3	IC3	2	Bottom elevation of each retention pond
1	ID	+	The id of the retention pond
2	HRBOTM(ID)	+	The bottom elevation of each retention pond.
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	IP	Card group identifier
C 3	IC3	1	Error tolerance for the simulation of retention ponds.
1	TOLBR	+	The error tolerance for the simulation of retention ponds.
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	RF	Card group identifier

COSFLOW.DOC - 164

C 3	IC3	1	Rainfall information
1	ID	+	The id of the retention pond.
2	ITYPON(ID)	+	Index of rainfall x-y series for this retention pond.
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	CB	Card group identifier
C 3	IC3	1	The intra-boundary information between retention ponds and subsurface region.
1	M	+	The subsurface global element number of this intra-boundary.
2	IDFACE	+	Element face id of this boundary.
3	ID	+	The id of the retention pond on which the intra-boundary locates.
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	OU	Card group identifier
C 3	IC3	1	The outlet information for each retention pond.
1	ID	+	The id of the retention pond.
2	NPID	+	Number of 2-D overland nodes receiving the water spilled out from the outlet.
3	IOUTYP(ID)	+	The index of x-y rating curve for the water through the outlet.
(new line)		+	The 2-D overland node numbers. In total, NPID nodes must be provided.
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	MN	Card group identifier
C 3	IC3	1	Minimum pressure head must be kept for the intra-boundary nodes.
1	IHMIN	0	Constant minimum pressure head
		1	Variable depth, (Not available)
2	HMIN	+	Value of the constant minimum pressure head.
<u>XY Card</u>		<u>X-Y Series Parameters</u>	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C 1-2	IC1	XY	Card group identifier
C 3	IC3	1	Generation of any x-y series function.
1	i	+	Index number for x-y series,
2	NPOINT(i)	+	The number of x-y values in the series.
3	DELTA X	0	Dummy values,
4	DELTA Y	0	Dummy values,
5	REPEAT	0	Dummy values,
6	BEGCYC	0	Dummy values,
7	TNAME	+	A character string representing the name of the XY series.
(new line)X,Y			After XY card, the x-y values of the series are listed one pair per line

up to NPOINT(i).

<u>EN Card</u>		<u>End of data control</u>	
<u>Field</u>	<u>Variable</u>	<u>Value</u>	<u>Description</u>
C1-2	IC1	EN	Card group identifier
C 3	IC3	D	End of input data

A.5 The formats of initial condition files and solution files

The output from COSFLOW consists of a printed output file and a number of solutions files. The printed output file is a text file listing a summary of the input, iteration and convergence data, and solution summaries. The solution files are used for post-processing or as initial conditions for subsequent runs. The content and format of the solution files are describes as follows.

The pressure head, moisture content (nodal), concentration, water depth of subsurface run-off, and water surface elevation of retention ponds and canals are saved to files using scalar GMS data set file format. The 3-D velocity field and 2-D overland flow field are saved to files with vector data set file format. With the scalar data sets, one scalar value is listed per node, in sequential order based on the node IDs. If the data set is transient, a complete set of scalar values is listed for each time step. With vector data sets, the x, y, and z components of the vector (velocity in this case) are listed on a node by node basis.

Data set files can be saved in either text or binary format. The binary format results in smaller files and can result in large reductions in the disk space required, particularly when doing transient analysis on large meshes.

A summary of the text version of the scalar data set file format is shown in Figure A.5.

DATASET	/* File type identifier */
OBJTYPE type	/* Type of objects */
BEGSCL	/* Beginning of scalar data set */
ND numdata	/* Number of data values */
NC numcells	/* Number of cells or elements */
NAME "name"	/* Data set name */
TS istat time	/* Time step of the following data */
stat ₁	
stat ₂	
.	
.	
stat _{numcells}	
val ₁	
val ₂	
.	
.	
Val _{numdata}	
/* Repeat TS card for each time step */	
ENDDS	/* End of data set */

DATASET	/* File type identifier */
OBJTYPE type	/* Type of objects */
BEGVEC	/* Beginning of vector data set */
ND numdata	/* Number of data values */
NC numcells	/* Number of cells or elements */
NAME "name"	/* Data set name */
TS istat time	/* Time step of the following data */
stat ₁	
stat ₂	
.	
.	
stat _{numcells}	
VX ₁ VY ₁ VZ ₁	
VX ₂ VY ₂ VZ ₂	
.	
.	
VX _{numdata} VY _{numdata} VZ _{numdata}	
/* Repeat TS card for each time step */	
ENDDS	/* End of data set */

Figure A.5 Text Data Set File Format

The first line of the file is a card without any fields which serves as a file type identifier.

Card Type	DATASET
Description	File type identifier
Required	Yes

The next line is an identifier that tells GMS which of the of objects the data set is associated with. For COSFLOW, the object type should always be wither 3D mesh or 2D mesh.

Card Type	OBJTYPE		
Description	Identifies the type of objects that the data sets in the file are associated with.		
Required	YES		
Format	OBJTYPE type		
Sample	OBJTYPE mesh3d		
Field	Variable	Value	Description
1	type	tin mesh2d grid2d scat2d mesh3d grid3d scat3d	TINS 2D meshes 2D grids 2D scatter points 3D meshes 3D grids 3D scatter points

To begin a data set, either a BEGSCL or BEGVEC card is required, depending on the type of data set.

Card Type	BEGSCL
Description	Marks the beginning of a set of cards defining a scalar data set.
Required	YES

Card Type	BEGVEC
Description	Marks the beginning of a set of cards defining a vector data set.
Required	YES

The pair of lines are the ND and NC cards. These cards are used to specify the number of nodes and elements in the mesh.

Card Type	ND		
Description	The number of data values that will be listed per time step. This number should correspond to the number of nodes for a 3D mesh or 2D mesh.		
Required	YES		
Format	ND numdata		
Sample	ND 10098		
Field	Variable	Value	Description
1	numdata	> 0	The number of items. At each time step, numdata values are listed.

Card Type	NC		
Description	This number should correspond to the number of elements for a 3D mesh or 2D mesh.		
Required	YES		
Format	ND numcells		
Sample	ND 3982		
Field	Variable	Value	Description
1	numdata	> 0	The number of elements.

The next line is for the name of data set.

Card Type	NAME		
Description	The name of the data set		
Required	YES		
Format	NAME "name"		
Sample	NAME "Total head"		
Field	Variable	Value	Description
1	"name"	string	The name of the data set, in double quotes.

A data set can contain multiple solutions, each solution representing a complete set of nodal values at a particular time step. The TS card is used to list a time value and the corresponding set of scalar or vector values.

If the solution is steady state, only one TS card is used and the time value is set to 0.0. If the solution is transient, the TS card is repeated once for each time step.

Card Type	TS (SCALAR)		
Description	Defines a set of scalar values associated with a time step. It should be repeated for each time step.		
Required	YES		
Format	TS istat time stat ₁ stat ₂ . . stat _{numcells} val ₁ val ₂ . . Val _{numdata}		
Sample	TS 0 12.5 34.5 74.3 58.4 72.9		
Field	Variable	Value	Description
1	istat	0	Use status flags from previous time step. For the first time step, this value indicates that all cells are active. Status flags will be listed.
		1	
2	time	> 0	The time step value. This number is ignored if there is only one time step.
	stat	0	Inactive
		1	Active
			One status flag should be repeated pre line for each cell or element. These flags are included only when ISTAT = 1.
	val	±	The scalar value, one pre line.

Card Type	TS (VECTOR)
-----------	-------------

Description	Defines a set of vector values associated with a time step. It should be repeated for each time step.		
Required	YES		
Format	TS istat time stat ₁ stat ₂ . . stat _{numcells} vx ₁ vy ₁ vz ₁ vx ₂ vy ₂ vz ₂ . . vx _n vy _n vz _n		
Sample	TS 0 12.5 34.5 74.4 34.4 74.3 643.4 36.3 58.4 745.4 46.3 72.9 734.3 345.3		
Field	Variable	Value	Description
1	istat	0 1	Use status flags from previous time step. For the first time step, this value indicates that all cells are active. Status flags will be listed.
2	time	> 0	The time step value. This number is ignored if there is only one time step.
	stat	0 1	Inactive Active One status flag should be repeated pre line for each cell or element. These flags are included only when ISTAT = 1.
	vx vy vz	±	The vector values, one set pre line.

Each data set should be terminated with an ENDDS card.

Card Type	ENDDS
Description	Signals the end of a set of cards defining a data sest
Required	YES

APPENDIX B: NUMERICAL FORMULATION

The initial-boundary value problem described by the governing equations of the 3-D subsurface flow module of COSFLOW with the boundary conditions cannot, in general, be solved analytically using current applied mathematics. Hence, in order to solve these sets of governing equations, numerical methods are the only mathematical tools capable of handling this task. Although there are many different numerical approximation methods capable of reducing partial differential equations to simpler systems of algebraic equations, there are only two numerical methods that are most common and that can be employed to the most basic form of the governing equations. These two numerical methods are the finite difference and finite element methods. The basic difference between these two methods is that the finite element method is based upon approximating the function, while the finite difference method is founded upon approximating the derivatives of the function. Therefore, the finite difference method only produces solutions at discrete points, while the finite element method yields spatially continuous solutions. Also, the finite element method offers numerous advantages over the finite difference method, such as (a) anisotropy and heterogeneity of aquifers are easily taken care of, (b) no need to formulate special formulae to incorporate irregular boundaries, (c) computer storage and computational time can sometimes be saved because often less nodal points are needed to portray the region of interest to the same level of accuracy, (d) irregular grids for handling different levels of spatial discretization in different sections of the region of interest can be incorporated, and lastly, (e) the integral formulation used in this method permits the flux types of the boundary conditions to come about naturally [Yeh, 1987]. Thus, the finite element method is used for the spatial discretization in this model. The theoretical background as well as numerical procedures of this method can be found in any good finite element method book, such as Istok [1989], and therefore will not be described here. A brief summary of the numerical procedure for applying the finite element method can be found in Yeh [1987].

Each flow module in COSFLOW includes 4 options for solving the finite element equations. In other words, there are four iteration methods (block iteration/direct solver, successive point iteration, polynomial

preconditioned conjugate gradient, and incomplete Cholesky preconditioned conjugate gradient methods) for solving the linearized matrix equations. Direct elimination methods are not used for solving 3-D problems in this report because it is impractical to deal with large three-dimensional problems. Because the Newton-Raphson method will yield a non-symmetric matrix, the Picard method is used to linearize the matrix equation.

To handle a large variety of possible problem sets, the 3-D subsurface flow module contains 16 optional numerical schemes. Specifically, the mixture of schemes includes the combinations of: (a) the 4 options for solving the resulting matrix equation as mentioned in the above discussion, (b) two options (lumping and consistent) for handling the mass matrix resulting from the storage term, and (c) two options (time-weighted difference and mid-difference) for approximating the time derivatives. The theoretical background for (b) and (c) may also be found in any respectable matrix computation book and in Yeh [1991].

On the other hand, the 1-D channel flow module involves only the concept of water budget. Its associated governing equation is a 1-st order ordinary differential equation, with respect to time, for each channel reach and therefore can be solved with given initial conditions. For any channel reach, we first evaluate incoming and outgoing volumetric flow rates from a geometric point of view: precipitation and evaporation (through the top boundary), discharge through gates or weirs (through upstream/downstream boundaries), interaction with subsurface systems (through side and bottom boundaries), and pumping (internal source/sink). And then we compute for the new water stage for the channel reach under the user-specified constraint of head change during a channel time step. Such constraint is designed for accuracy.

To account for the interaction between 3-D subsurface and 1-D channel flow systems, we calculate volumetric flow rate through each subsurface/channel interface segment during each channel time step, and then evenly distribute this flux to the nodes of the interface segment. These nodes will serve as point sources/sinks in computing subsurface flow. The strength of such a point source/sink is the accumulated volumetric flow rate at that node through a number of channel time steps that are included in a subsurface time step span. The

contribution to the accumulated flux from a channel time step is equal to the multiplication of the volumetric flow rate and the size of the channel time step divided by the size of subsurface time step that cover the channel time step.

B.1 Numerical Approximation of the 1-D Channel Water Budget Equation

In COSFLOW, the governing equation of 1-D channel flow is derived based on the concept of water budget. It can be written as follows.

$$B(HRT(i)) \frac{dHRT(i)}{dt} = QT(i) + QS(i) + QUD(i) + QP(i) \quad i \in [1, NORH] \quad (B.1)$$

where $HRT(i)$ is the water stage of channel reach i ; $B(HRT(i))$ is the top surface area of channel reach i when the water stage of channel reach i is $HRT(i)$; $QT(i)$ is the volumetric flow rate through the top boundary, due to precipitation and/or evaporation, of channel reach i ; $QS(i)$ is the volumetric flow rate through the side and bottom boundary, due to the interaction between subsurface and channel flow systems, of channel reach i ; $QUD(i)$ is the volumetric flow rate through the upstream/downstream boundary, due to discharges of gates and/or weirs, of channel reach i ; $QP(i)$ is the volumetric flow rate of internal source/sink, due to pumping, of channel reach i .

All the volumetric flow rates are positive if water is directed into channel reaches and negative if water is going out. $B(HRT(i))$ can be computed by the following equation.

$$B(HRT(i)) = \sum_{j=1}^{ncer(i)} BW(i,j) + (s1(i,j) + s2(i,j)) (HRT(i) - BE(i,j)) \quad i \in [1, NORH] \quad (B.2)$$

where $ncer(i)$ is the number of channel elements included in channel reach i ; $BW(i,j)$ is the bottom width of the j -th channel element in channel reach i ; $BE(i,j)$ is the bottom elevation of the j -th channel element in channel reach i ; $s1(i,j)$ is the right side slope of the j -th channel element in channel reach i ; $s2(i,j)$ is the left side slope of the j -th channel element in channel reach i . After mathematical manipulation, Eq. (B.2) can be written as

$$B(HRT(i)) = C(i) + D(i)HRT(i) \quad i \in [1, NORH] \quad (B.3)$$

where $C(i)$ and $D(i)$ are constant to $HRT(i)$. Substituting Eq. (B.3) into Eq. (B.1) yields

$$(C(i) + D(i)HRT(i)) \frac{dHRT(i)}{dt} = Q(i) \quad i \in [1, NORH] \quad (B.4)$$

Given $HRT(i)$ initially, Eq. (B.4) can be solved one channel time step by one channel time step. For accuracy, the change of $HRT(i)$ during one channel time step can be limited by users. Therefore, a smaller time step size than the specified channel time step size may be needed. This will be performed by the computer code. As a result, more than the given channel time steps would be included in one subsurface time step.

B.1.1 Computation of Interaction Between Subsurface and Channel Flow Systems

In COSFLOW, the subsurface/channel interfaces can be conceptualized as lines, vertical planes, or planes describing the true interface relative to the 3-D subsurface domain. The computation of the flux through the interface is achieved as follows.

(1). For the case that the interfaces are treated as lines:

The flux through the interface of any channel element can be computed by

$$QS(i,j) = S(i,j)K(i,j) \frac{HS(i,j) - HRT(i)}{d(i,j)} \quad (B.5)$$

where $QS(i,j)$ is the flux flowing into the j -th channel element of channel reach i ; $S(i,j)$ is the area of channel bottom and side surface under channel water of the j -th channel element of channel reach i ; $K(i,j)$ and $d(i,j)$ are the perimeter conductivity and thickness associated with the j -th channel element of channel reach i ; $HS(i,j)$ is the subsurface total head associated with the j -th channel element of channel reach i . This subsurface total head can be calculated by

$$HS(i,j) = \frac{HS(i,j,1) + HS(i,j,2)}{2} \quad (B.6)$$

where $HS(i,j,1)$ and $HS(i,j,2)$ are the total heads of the 1-st and 2-nd subsurface nodes associated with the j -th

channel element of channel reach i. $S(i,j)$ can be calculated as

$$S(i,j) = L(i,j) \{BW(i,j) + [s1(i,j) + s2(i,j)][HRT(i) - BE(i,j)]\} \quad (B.7)$$

where $L(i,j)$ is the length of the j-th channel element of channel reach i.

By summing $QS(i,j)$ over j, the flux through the interface associated with channel reach i, $QS(i)$, can be obtained. Additionally, we distribute $QS(i,j)$ evenly to the two subsurface nodes associated with the j-th channel element of channel reach i, so that the flux through the interface can be treated as point sources/sinks to the 3-D subsurface system. The contribution of source/sink strength from $QS(i,j)$ to either one of the two subsurface nodes is

$$Q_{1(or2)} = \frac{QS(i,j)}{2} \frac{\Delta t_c}{\Delta t_s} \quad (B.8)$$

where Q_1 is the contribution to the 1-st subsurface node associate with the j-th channel element of channel reach i; Δt_c is the channel time step size truly used for the current channel time step; Δt_s is the subsurface time step size.

(2). For the case that the interfaces are treated as vertical plane:

In this case, we need to specify the bottom line segments of vertical planes as channel elements (in IFS card of 1-D canal model file) and specify vertical planes using vertical plane elements with 3 or 4 subsurface nodes as the element nodes for each plane element (in IFS card of 1-D canal model file). With the bottom line segment of vertical planes specified as channel elements, we are able to compute flux through the bottom boundary of each channel element by using Eq. (B.5). Here $S(i,j)$ includes only the bottom surface of the j-th channel element of channel reach i. On the other hand, we compute flux through the side boundary one plane element by one plane element. For each plane element, we first determine the velocity associated with each element node. If the node is below the channel water, the following equation is used.

$$VS(i,j,k) = K(i,j) \frac{HS(i,j,k) - HRT(i)}{d(i,j)} \quad (B.9)$$

where $VS(i,j,k)$ is the velocity computed at the k-th element node of the j-th plane element in reach i; $HS(i,j,k)$

is the total head on the surface associated with the k-th element node of the j-th plane element in reach i.

If the node is above the channel water, the velocity is zero if the subsurface pressure head associated with the node is negative, whereas the velocity can be computed by the following equation when the associated subsurface pressure head is positive.

$$VS(i,j,k) = K(i,j) \frac{HS(i,j,k)}{d(i,j)} \quad (B.10)$$

Based on the above, we can compute the average velocity for a plane element. Multiplying this velocity by the surface area of the plane element yields the volumetric flow rate through the plane element. Such a volumetric flow rate has to be further multiplied by a factor of 2 because vertical planes represent two channel sides in real. By summing up all the contributions through the bottom and side boundaries of reach i, $QS(i)$ is thus determined. Like that in (1), we can evenly distribute volumetric flow rate of either a line element (for the bottom boundary) or a plane element (for the side boundary) to its element nodes, such that the interface fluxes can be taken into account by point sources/sinks in the computation of subsurface flow.

(3). For the case that planes describing the true interface are used:

In this case, only computation regarding the side boundary of (2) needs to be implemented. It is thus not necessary to repeat.

B.2 Numerical Approximation of the 2-D Overland Systems

In COSFLOW, the 2-D flow module includes retention ponds and overland systems. The number of retention ponds can be specified as any number and those are confined by dikes. The retention ponds are described by water budget equations, whereas the dikes may be discretized to 3-D subsurface regime. On the top of each retention pond, an outlet is designed to discharge the water to the overland flow region via spillway, which is considered point sources for 2-D system. Therefore, the interaction between 2-D overland flow and 3-D subsurface systems and that between retention ponds and 3-D subsurface systems must be taken into account.

Regarding the connection between retention ponds and 1-D canal system, the amount of volumetric flow rate via pumping from canals is a source for the retention pond system.

The following sections describe the numerical formulations of 2-D overland flow and retention ponds systems, respectively. The computation of interactions between 3-D subsurface flow and 2-D overland flow systems, and between 3-D subsurface flow and retention ponds systems plays a very important role in the model and will be described afterwards.

B.2.1 Numerical Approximation of 2-D Overland Flow Equation

The governing equation is written as

$$\frac{\partial h}{\partial t} + \nabla \cdot (\mathbf{V}h) = r - i + q \quad (\text{B.11})$$

where h is water depth; \mathbf{V} is a velocity vector; q is the flux rate from retention ponds, r is rainfall rate, and i is infiltration rate. To solve the above nonlinear partial differential equation, the Galerkin finite element method is employed to discretize the system and the Picard method is used to linearize the matrix equation. To handle a large variety of possible problem sets, the 2-D overland flow module for COSFLOW contains 16 optional numerical schemes. Specifically, the mixture of schemes includes the combinations of: (a) the 4 options (direct solver, successive point iteration, polynomial preconditioned conjugate gradient, and incomplete Cholesky preconditioned conjugate gradient methods) for solving the linearized matrix equations, (b) two options (lumping and consistent) for handling the mass matrix resulting from the storage term, and (c) two options (time-weighted difference and mid-difference) for approximating the time derivatives. The theoretical background for (b) and (c) may also be found in any respectable matrix computation books and in Yeh [1991].

B.2.1.1 Spatial discretization with the Galerkin finite element method

When using the finite element method, the water depth is approximated by:

$$h \approx \hat{h} = \sum_{j=1}^N h_j(t) N_j(x, y, z) \quad (B.12)$$

where h_j and N_j are the amplitude of h and the base function, respectively, at nodal point j and N is the total number of nodes. After defining a residual and forcing the weighted residual to zero, the flow equation, Eq. (B.11), is approximated as:

$$\left[\int_R N_i N_j dR \right] \frac{dh_j}{dt} + \left[\int_R K \nabla N_i \cdot \nabla N_j dR \right] h_j = \int_R N_i (q + r - i) dR - \int_R K \nabla N_i \cdot \nabla z dR + \int_B K \mathbf{n} \cdot (\nabla h + \nabla z) dB \quad (B.13)$$

In matrix form, Eq. (B.13) is written as:

$$[M] \left\{ \frac{dh}{dt} \right\} + [S] \{h\} = \{Q\} + \{G\} + \{B\} \quad (B.14)$$

where $\{dh/dt\}$ and $\{h\}$ are the column vectors containing the values of dh/dt and h , respectively, at all nodes; $[M]$ is the mass matrix resulting from the storage term; $[S]$ is the stiff matrix resulting from the action of conductivity; $\{Q\}$, $\{G\}$, and $\{B\}$ are the load vectors from the internal source/sink, gravity force, and boundary conditions, respectively. Furthermore, the mass matrix, $[M]$, and stiff matrix, $[S]$, are described as:

$$M_{ij} = \sum_{e \in M_e} \int_{R_e} N_\alpha^e N_\beta^e dR \quad (B.15)$$

and

$$S_{ij} = \sum_{e \in M_e} \int_{R_e} K \nabla N_\alpha^e \cdot (\nabla N_\beta^e) dR \quad (B.16)$$

where R_e is the region of element e , M_e is the set of elements that have a local side α - β coinciding with the global side i - j , and N_α^e is the α -th local base function of element e .

In addition, the three load vectors, $\{Q\}$, $\{G\}$, and $\{B\}$, are described as:

$$Q_i = \sum_{e \in M_e} \int_{R_e} N_\alpha^e (q + r - i) dR \quad (B.17)$$

$$G_i = - \sum_{e \in M_e} \int_{R_e} K \nabla N_\alpha^e \cdot \nabla z dR \quad (B.18)$$

and

$$B_i = - \sum_{e \in N_{sc}} \int_{B_e} -K N_\alpha^e \mathbf{n} \cdot (\nabla h + \nabla z) dB \quad (B.19)$$

where N_{sc} is the set of boundary segments that have a local node α coinciding with the global node i , and B_e is the length of boundary segment e .

In most finite element work, the flow velocity components given in Eq. (2.18) are calculated numerically by taking the derivatives of the simulated h as

$$\mathbf{V} = -\frac{K}{h} ((\nabla N_j) h_j + \nabla z) \quad (B.20)$$

The above formulation results in velocity field which is not continuous at element boundaries and nodal points if the variation of h is other than linear or constants. The alternative approach would be to apply the Galerkin finite element method to Eq. B.20, thus one obtains

$$[T] \{V_x\} = \{D_x\} \quad (B.21a)$$

$$[T] \{V_y\} = \{D_y\} \quad (B.21b)$$

where the matrix $[T]$ and the load vectors $\{D_x\}$ and $\{D_y\}$ are given by

$$T_{ij} = \sum_{e \in M_e} \int_{R_e} N_\alpha^e N_\beta^e dR \quad (B.22a)$$

$$D_{xi} = - \sum_{e \in M_e} \int_{R_e} N_\alpha^e K i \cdot (\nabla h + \nabla z) dB \quad (B.22b)$$

$$D_{yi} = - \sum_{e \in M_e} \int_{R_e} N_\alpha^e K j \cdot (\nabla h + \nabla z) dB \quad (B.22c)$$

where V_x and V_y are the flow velocity components along the x- and y-directions, respectively and \mathbf{i} and \mathbf{j} are the unit vector along the x- and y-coordinates, respectively.

The reduction of the partial differential equation Eq. (B.11) to the set of ordinary differential equations Eq. (B.13) simplifies the evaluation of integrals on the right hand side of Eqs. (B.15) through (B.19) for every boundary segment e . The major tasks that remain to be done are the specification of base and weighting functions and the performance of integration to yield the element matrices. Linear hexahedral elements are employed in this documentation.

B.2.1.2 Base and weighting functions

The construction of base functions for quadrilateral elements is best accomplished using the local coordinates (ξ, η) . In the local coordinates, the original quadrilateral element is mapped into a square whose corners are located at $\xi = \pm 1$ and $\eta = \pm 1$ as shown in Figure Figure B.1.

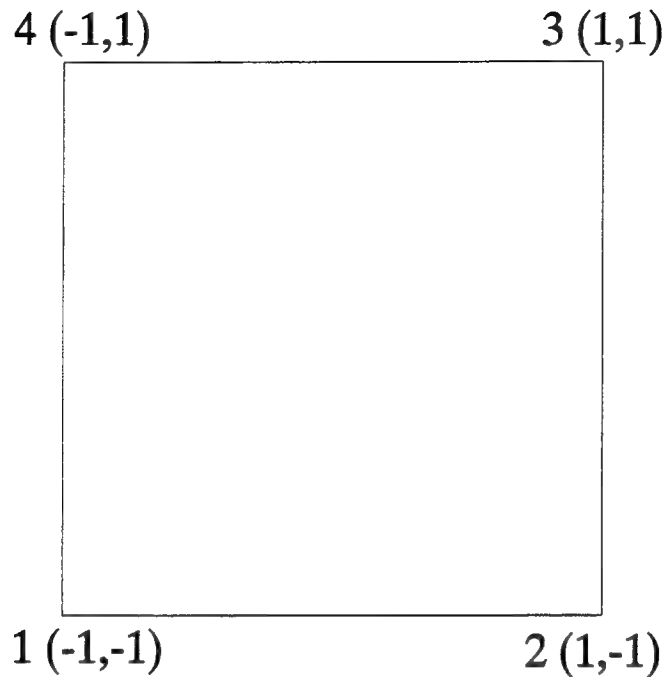


Figure B.1 A quadrilateral element in local coordinate

For bi-linear quadrilateral elements, the four base functions are obtained by taking the tensor product of the three base functions of the linear line elements as

$$N_i(\xi, \eta) = \frac{1}{4}(1 + \xi \xi_i)(1 + \eta \eta_i), \quad i = 1, 2, \dots, 4 \quad (\text{B.23})$$

Because the Galerkin finite element method is used to solve the flow equations, the set of 4 weighting functions is taken as the same set of 4 base functions.

B.2.1.3 Numerical integration

To complete the reduction of the partial differential equation [Eq. (B.11)] to the ordinary differential equation [Eq. (B.13)], one has to evaluate the integrals on the right-sides of Eqs. (B.15) through (B.19) for every element to yield the element mass matrix $[M^e]$ and the stiff element matrix $[S^e]$ as well as the element gravity column vector $\{G^e\}$, the source/sink column vector $\{Q^e\}$, and the boundary column vector $\{B^e\}$ as

$$M_{\alpha\beta}^e = \int_{R_e} N_{\alpha}^e N_{\beta}^e dR \quad (B.24)$$

$$S_{\alpha\beta}^e = \int_{R_e} K \nabla N_{\alpha}^e \cdot \nabla N_{\beta}^e dR \quad (B.25)$$

$$Q_{\alpha}^e = \int_{R_e} N_{\alpha}^e (q + r - i) dR \quad (B.26)$$

$$G_{\alpha}^e = - \int_{R_e} K \nabla N_{\alpha}^e \cdot \nabla z dR \quad (B.27)$$

and

$$B_{\alpha}^e = - \int_{B_e} N_{\alpha}^e K \mathbf{n} \cdot (\nabla h + \nabla z) dB \quad (B.28)$$

Since Eqs. (B.24) through (B.28) are written in the global coordinate and the base functions are defined in the local coordinate, a transformation between the global and local coordinate is needed. The required transformation from global coordinate to local coordinate is obtained by via the base functions as

$$x = \sum_{j=1}^4 x_j N_j(\xi, \eta) \quad (B.29)$$

$$y = \sum_{j=1}^4 y_j N_j(\xi, \eta) \quad (B.30)$$

Because the coordinate transformation uses the base functions, the element is termed the "isoparametric" element.

Using the transformation in Eqs. (B.29) and (B.30), we convert the differentiation of the base function with respect to the global coordinate to that with respect to the local coordinate by

$$\begin{Bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \end{Bmatrix} = [J]^{-1} \begin{Bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \end{Bmatrix}$$

(B.31)

$$[J] = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix} \quad [J]^{-1} = \text{Inverse of } [J]$$

where $[J]$ is the Jacobian of the transformation. In the mean time, a differential area written in the local coordinate becomes

$$\int_e dR = \int_{-1}^1 \int_{-1}^1 J d\xi d\eta \quad (B.32)$$

With Eqs. (B.31) and (B.32), all the integrals in Eqs. (B.24) through (B.28) can be reduced to the following form

$$\int_{-1}^1 \int_{-1}^1 f(\xi, \eta) J d\xi d\eta \quad (B.33)$$

the integration of which can easily be carried out with a $2 \times 2 = 4$ point Gaussian quadrature. The line integration of Eq. (B.33) is straightforward in two-dimensional space.

B.2.2 Numerical Approximation of Retention Pond Water Budget Equation

In COSFLOW, the governing equation of retention ponds is derived based on the concept of water budget. It can be written as follows.

$$A(j) \frac{\partial H(j)}{\partial t} = QT(j) + QP(j) - QU(j) + QS(j) \quad j \in [1, N] \quad (B.34)$$

where $A(j)$ is the water surface area of the j -th retention pond when the water surface elevation is $H(j)$; $H(j)$ is the water surface elevation of the j -th retention pond; $QT(j)$ is the net volumetric flow rate into the j -th retention pond due to precipitation/evaporation; $QP(j)$ is the net volumetric flow rate into the j -th retention pond due to pumping; $QU(j)$ is the net volumetric flow rate out of the j -th retention pond through the spillway; $QS(j)$ is the net volumetric flow rate into the j -th retention pond from the subsurface; N is the number of retention pond in the region of interest. $QU(j)$ and $QS(j)$ are simulated similar to that in channel. $QU(j)$ depends only on water surface elevation of the retention pond, whereas $QS(j)$ depends on both water surface elevation of the retention pond and total head in subsurface media.

All the volumetric flow rates are positive if water is directed into retention ponds and negative if water is going out. $A(j)$ is the average of water surface areas at current and previous times. The water surface area at the previous time is calculated Eq. (B.35a).

$$A^p = RBAREA(j) + \frac{HRP(j) - HRBOTM(j)}{HMXPON(j) - HRBOTM(j)} (RPAREA(j) - RBAREA(j)) \quad (B.35a)$$

where $HRP(j)$ is the water surface elevation of the j -th retention pond at the previous time, $RBAREA(j)$ and $RPAREA(j)$ are the water surface areas at zero and full water depth, respectively of the j -th retention pond, $HRBOTM(j)$ is the bottom elevation of the j -th retention pond, and $HMXPON(j)$ is the allowed maximum water surface elevation of the j -th retention pond. Likewise, the water surface area at the current time is calculated based on the current water surface elevation, $HR(j)$, and expressed as Eq. (B.35b).

$$A^c = RBAREA(j) + \frac{HR(j) - HRBOTM(j)}{HMXPON(j) - HRBOTM(j)} (RPAREA(j) - RBAREA(j)) \quad (B.35b)$$

And the area $A(j)$ in Eq. (B.34) is equal to $(A^p + A^c)/2$. Substituting $A(j)$ into Eq. (B.34), we can obtain the following quadratic equation.

$$A_1 HRT(j)^2 + A_2 HRT(j) + A_3 = 0 \quad (B.36)$$

where the coefficients A_1 , A_2 , and A_3 are

$$\begin{aligned}
A_1 &= \text{RPAREA}(j) - \text{RBAREA}(j) \\
A_2 &= 2(\text{RBAREA}(j) \times \text{HMXPON}(j) - \text{HRBOTM}(j) \times \text{RPAREA}(j)) \\
A_3 &= -\text{HRP}(j)^2 \times A_1 - \text{HRP}(j) \times A_2 - 2Q(j)(\text{HMXPON}(j) - \text{HRBOTM}(j)), \\
\text{where } Q(j) &= \text{QT}(j) + \text{QP}(j) - \text{QU}(j) + \text{QS}(j)
\end{aligned}
\tag{B.37}$$

Then, the current water surface elevation of the j -th retention pond can be thus obtained and given as follows.

$$\text{HR}(j) = \frac{-A_2 \pm \sqrt{A_2^2 - 4A_1A_3}}{2A_1}
\tag{B.38}$$

The decision for taking “+” or “-” sign in Eq. (B.38) is based on the value of $Q(j)$. Normally, it takes “+” sign. However, if $Q(j)$ is negative and the calculated $\text{HR}(j)$ is greater than $\text{HRP}(j)$, the “-” sign is the correct one in Eq. (B.38) and Eq. (B.38) is recalculated using the “-” sign. The computation of $\text{QT}(j)$, $\text{QP}(j)$, and $\text{QU}(j)$ is straightforward. The computation of $\text{QS}(j)$ is explained in B.2.4.

B.2.3 Computation of Interaction between Subsurface Flow and Overland Flow System

The mechanism for coupling 2-D overland and 3-D subsurface systems is basically under the assumption of weak coupling. The excess rainfall, which is defined as the normal value of the rainfall rate plus the ponding depth per unit time, is to sink to the subsurface first. The amount of water that the subsurface cannot absorb will become surface runoff. With this conceptualization, there can be two types of boundary conditions implemented on the ground surface for the computation of subsurface flow. One is the Dirichlet type which has the head value as the same as the associated water depth on the overland, and the other one is the flux type which is equal to the excess rainfall rate. A subroutine named BCTOP is coded to determine the type of boundary condition to apply to the ground surface. A nonlinear loop is required to achieve correct implementation of boundary conditions. If the computed result is consistent with the guessed boundary condition, i.e., the number of nodal points needed to switch boundary conditions is zero, the final solution has been thus obtained. Then, the infiltration rate is the rainfall rate at the location with the flux type boundary condition implemented or is calculated according to

Darcy's law based on the pressure head distribution in the subsurface at where Dirichlet boundary conditions apply. This infiltration rate is the value i in Eq. (2.16).

B.2.4 Computation of Interaction between Subsurface Flow and Retention Pond Water Budget

The procedure to determine the flux, $QS(j)$ through interfaces between the subsurface and retention ponds is similar to that between the subsurface and overland systems, except all the interfacial nodes that are associated with one retention pond are considered to be of the same water depth. This is because of water budget for governing the retention pond. For simplification, the dike can be excluded in the discretization of 3-D domain and only the downward flux through the bottom face of retention pond needs to be considered. In practice, the dike needs to be discretized to a number of elements vertically as well as horizontally, and should be considered to be a part of 3-D subsurface domain. In this case, both the downward flux through the bottom face of retention pond and the fluxes through the side faces between the retention pond and dike need to be evaluated for infiltration.

BIBLIOGRAPHY

- Kolkman, P. A., Discharge relationships and component head losses for hydraulic structures, *Discharge Characteristics*, edited by D. S. Miller, BHR Group Limited, Bedford, UK, 1994.
- Huyakorn, P. S., E. P. Springer, V. Guvanasen, and T. D. Wadsworth, A three-dimensional finite-element model for simulating water flow in variably saturated porous media, *Water Resour. Res.*, 22(13), 1790-1808, 1986.
- Istok, J. D., Groundwater Modeling by the finite element method, American Geophysical Union, Washington D.C., pp. 495, 1989.
- van Genuchten, M. Th., A closed form equation for predicting the hydraulic conductivity of unsaturated soils, *Soil Science Society of America Journal*, 44, 892-898, 1980.
- Yeh, G. T., FEMWATER: A finite element model of water flow through saturated-unsaturated porous media, First Revision, Rep. ORNL-5567/R1, Oak Ridge Nat. Lab., Oak Ridge, Tenn., 37831, 258 pp., 1987.
- Yeh, G. T., Class notes: CE597D: Computational Subsurface Hydrology Part I, The Pennsylvania State University, University Park, Pa., 16802, 361 pp., Fall semester 1991.
- Yeh, G. T., J. R. Cheng, and H. P. Cheng, 3DFEMFAT: Users' Manual of a 3-Dimensional Finite Element Model of Flow And Transport through Saturated-Unsaturated Media, 1994.
- Yeh, G. T., H. P. Cheng, and H. C. Lin, 3DFEMFAT ver. 2C.0: Users' Manual of a 3-Dimensional Finite Element Model of Flow And Transport through Saturated-Unsaturated Media, 1995.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE September 1997	3. REPORT TYPE AND DATES COVERED Final report
4. TITLE AND SUBTITLE COSFLOW: A Finite Element Model Coupling One-Dimensional Canal, Two-Dimensional Overland, and Three-Dimensional Subsurface Flow		5. FUNDING NUMBERS	
6. AUTHOR(S) Gour-Tsyh (George) Yeh, Jing-Ru (Ruth) Cheng, Ming-Hsu Li, Hwai-Ping (Pearce) Cheng, Hsin-Chi J. Lin		8. PERFORMING ORGANIZATION REPORT NUMBER Technical Report CHL-97-20	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Department of Civil and Environmental Engineering, Pennsylvania State University, University Park, PA 16802, and U.S. Army Engineer Waterways Experiment Station, 3909 Halls Ferry Road, Vicksburg, MS 39180-6199		10. SPONSORING/MONITORING AGENCY REPORT NUMBER	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) U.S. Army Engineer District, Jacksonville P.O. Box 4970, Jacksonville, FL 32232-0019			
11. SUPPLEMENTARY NOTES Available from National Technical Information Service, 5285 Port Royal Road, Springfield, VA 22161.			
12a. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release; distribution is unlimited.		12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) <p>This report presents the user's manual of COSFLOW, which is a three-dimensional Finite Element Model coupling one-dimensional (1-D) Canal, two-dimensional (2-D) Overland, and three-dimensional (3-D) Subsurface FLOW. The 1-D channel flow is described by water budget under the assumption that equilibrium within each channel reach is achieved instantaneously. The 2-D overland flow is modeled with a diffusive wave approach. Retention ponds included in the overland flow are simulated with a water budget approximation. In this version, solute transport is not considered in the coupled system; but for the future consideration, the subsurface solute transport module is included in the computer code. In this report, however, emphasis is given to the coupling of flow between 3-D subsurface, 1-D channel, and 2-D overland/retention pond systems. The details associated with solute transport in the subsurface, such as density-dependent transport and salt intrusion, can be found in the user's manual of 3DFEMFAT.</p> <p>In the channel module, the volumetric flow rates from the channel to the subsurface nodes, or vice versa, are computed within each channel time-step, and are accumulated within each subsurface time-step (one subsurface time-step usually includes many channel time-steps). The pumping from the channel can be directed to another channel reach, a subsurface node, or a retention pond, which is designed to store excess water in a canal. The water level in retention ponds is calculated</p> <p style="text-align: right;">(Continued)</p>			
14. SUBJECT TERMS Darcy's velocity Galerkin finite element method Interaction between overland and subsurface flow Manning equation		One-dimensional channel flow Overland flow Richard's equation Subsurface flow	15. NUMBER OF PAGES 198 16. PRICE CODE
17. SECURITY CLASSIFICATION OF REPORT UNCLASSIFIED	18. SECURITY CLASSIFICATION OF THIS PAGE UNCLASSIFIED	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT

13. (Concluded).

based on mass balance between inflow such as rainfall, injection water from pumps, seepage from the surrounded dikes; and outflow, e.g., infiltration, seepage to the dikes, and water flowing out through outlets. In the overland flow module, the Galerkin finite element method is used to solve the diffusion type continuity equation. The infiltration rate is calculated based on the water depth on the ground and the pressure head distribution in the subsurface. The time-step size for overland flow is prescribed by users and could be several times the canal time-step size. The volumetric flow rate from surface runoff to the subsurface media, or vice versa, is computed in each overland time-step and is accumulated in each subsurface time-step. The accumulated fluxes thus play the roles of point sources/sinks in the subsurface module. After solutions for all the modules are obtained, the convergence of coupling is examined by comparing the infiltration rate of two successive iterations. If convergence is performed, the computation is achieved. Otherwise, another iteration is proceeded as desired.

For each site-specific application, 151 control-integers must be assigned using the parameter statement in the including file. In addition, if the material properties, the discharge rate from gates, and the discharge rate from weirs are specified by analytical functions, the subroutines SPROP3, QGATE, and QWEIR, respectively, must be modified by the users.

Input to the program includes the control indices, properties of the media either in tabular or analytical form, the geometry in the form of elements and nodes, and boundary and initial conditions either in a tabular form or subject to a constant value. Principal output includes the spatial distribution of pressure head, total head, moisture content, and Darcy velocity components for the subsurface system, and water stage for the channel system at any desired subsurface time-step. Fluxes through various types of boundaries of the subsurface system also are output. In the overland regime, the water depth, water surface elevation, and velocities in the x- and y-directions are also printed out at specific time-steps. Diagnostic variables such as the number of nonconvergent nodes and residuals may be printed if desired for debugging purposes.